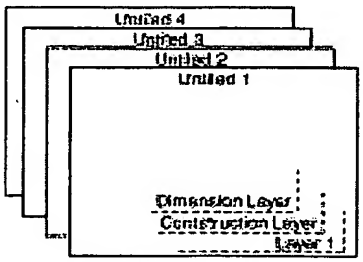
File Management

Whenever you open this Designer Elements program or choose *New* from the File menu, a new document appears. You can open multiple files to work with as desired.

Each file supports the layers feature. This feature adds a great deal of flexibility to your documents. Layers allow you to see various parts of a document individually or as part of the whole. Each document is made up of layers which can be hidden or displayed as needed.



Designer Elements Files

Consider file organization early. This is particularly important if you are sharing files with other people.

The following topics are covered in this chapter.

- Using Documents, including starting this Designer Elements program, opening files, recent file list and saving

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- Files and Preferences
- Files and Locking
- File Size & Deleting
- Uninstalling this Designer Elements program on Windows

Using Documents

A drawing can be a simple part or a complex assembly. A new document opens as *Untitled* and remains untitled until you save it. Saving a drawing is explained later in this chapter.

File Use and Operation System Short Cuts

This Designer Elements program takes advantage of some features of your operating systems when working with files.

Windows

This Designer Elements program allows you to open files and start up the program within the *Windows Explorer* or other folders. It also supports the *Drag and Drop* functionality.

Windows Explorer and other Folders

You can open Designer Elements program files by double-clicking on the file icon in Windows Explorer, the Desktop or another folder. If this Designer Elements program is not running, the program launches first and then opens the file.

If you want to open multiple files, hold down the CTRL key to select the files. Press the right mouse button and select the *Open* command. Each file opens in a separate drawing. If this Designer Elements program is not running, the program launches first, before the files open.

Drag and Drop

This Designer Elements program gives you the ability to manipulate files by dragging and dropping them.

You can drag and drop files into this Designer Elements program to open the files.

- If you drag and drop one or more files into an open drawing, they are merged into the drawing.

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- If you drag and drop one or more files into this Designer Elements program when no files are open, each opens individually.
- If you hold down the SHIFT key before dropping multiple files into this Designer Elements program when no files are open, files are merged into a single new drawing.
- You can drag and drop one or more files onto the Designer Elements program desktop icon and the files open individually with the program open.

Macintosh

You have the ability to double-click on a Designer Elements program file to launch the program and open the file.

- If you drag and drop one or more files into an open drawing, they are merged into the drawing.
- If you hold down the SHIFT key before dropping multiple files into this Designer Elements program when no files are open, files are merged into a single new drawing.
- You can drag and drop one or more files onto the Designer Elements program desktop icon and the files open individually with the program open.

File Menu Commands

The File menu contains commands to manipulate documents, including, *New*, *Open*, *Close*, *Save* and *Revert*.

Referral:

Use the *File>Import* command if you want to use drawings of other format types, such as DXF, IGES, or SAT. See Chapter 35 for more information.

New - CTRL+N (Windows); z +N (Macintosh)

This command in the File menu creates a new Designer Elements program document. The new document has no name (the title bar shows *Untitled 1*), and is set with the default options, such as pen style or grid display.

If you open more than one new document, the subsequent documents are numbered sequentially until you name them by saving.

Open - CTRL+O (Windows); z +O (Macintosh)

This command in the File menu opens an existing Designer Elements program document. Using this command you can also open Graphite files. A progress bar appears as the file is opening. This is especially helpful for large files.

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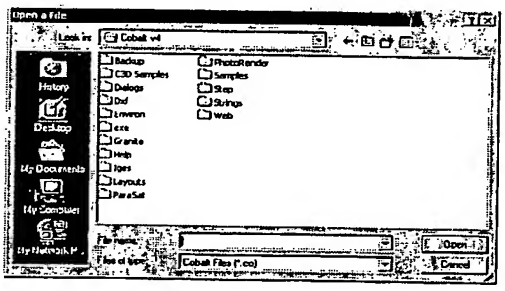
Tech Note:

If you want to open a Designer Elements program file in prior versions of Ashlar-Vellum software, export the file using the appropriate export version. This Designer Elements program embeds ACIS data in its binary files. Vellum Solids 99 and 98 use an earlier ACIS version and ACIS is not backwards compatible.

The document appears in the drawing area maintaining the same settings as the last time you saved it.

The dialog box allows you to specify the document you want and lets you change folders, if necessary.

Windows - If the *Open as read only* box is checked, the file can be opened and printed but not altered. Macintosh - If a file is locked, it can be opened and printed but not altered. You can unlock a file by selecting the file and choosing Get Info (Macintosh standard).



Opening a Document

1.Choose ***File>Open.***

The dialog box appears.

The current folder displays with the files and/or folders it contains.

2.Choose the appropriate folder containing the document you want to open.

3.Click the File name you want to open in the list box.

4. Click OK.

Recent File List

Another way to access files opened recently is through the *Recent File List* that appears in the File menu after the *Exit* command (Windows) or *Quit* command (Macintosh). This list contains the names and paths of the most recent files opened in this Designer Elements program.

The number of files displayed depends on the number selected in the Filing page of the Preferences dialog box. You can display as many as 20 files in this list.

To open a file from the *Recent File List*, select the file name from the File menu. If the file has been moved since it was last used and the path is no longer accurate, This Designer Elements program will ask you to locate the file by providing the standard Open dialog box.

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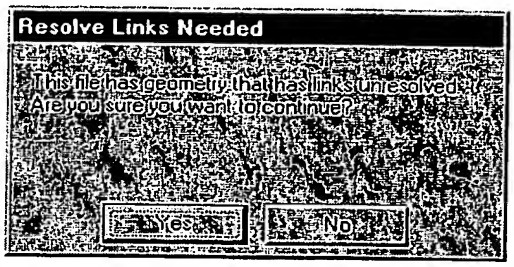
Saving a Drawing

You can save a drawing by choosing ***File>Save*** or ***Save As***. The file is stored on your computer in the folder you specify.

If you attempt to save a file containing links that are not resolved, the following warning appears:



Save - CTRL+S (Windows); z +S (Macintosh)

This command in the File menu saves the current Designer Elements program document to its original folder. If you want to save it to a different folder or with a different name, choose *Save As*. If you choose *Save* and the document has not been saved previously, the Save As dialog box appears automatically, allowing you to name the document and specify the folder in which to save it.

If you don't make a geometrical change to any object in your file, the *Save* command is not available. Changing an object's layer is not recognized as a geometrical change. If you have named and saved the document before, a brief message appears when you choose *Save*. The program pauses while it updates the information.

Note: You should save frequently. It is important to save before performing any intricate, multistep procedure. That way, if the result is not exactly what you had hoped, you can abandon the file by closing it without saving.

Save As

This command in the File menu saves the current document. A dialog box appears so you can name the current document, give it a different name, or save it to a different directory. You can use the *Save As* command to make a backup of a document.

(Windows) You also have the ability of saving a file as a Read only file. Click the check box for that option.

Tech Note:

If you try to save a file with the name of a locked file, a warning message will appear.

If you delete objects before saving a file, the data for that information may still be retained with the file. Perform the delete operation again to guarantee the data is removed.

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Saving in the Current Directory

1. Choose ***File>Save As***.

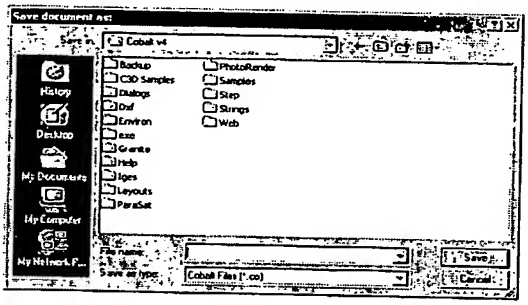
   The Save document as dialog box appears.

2. If necessary, display a different folder.

3. Type the name you want to use in the File name box.

4. Click OK.

   The filename extension *.vs* is automatically appended to the filename.



Tech Note:

Macintosh users: Although the .co, .xe, ar, .ne and .vs extension is not required for the file name, it is included to ease file transfer between Macintosh and Windows.

Making a Backup

You should make a backup of your work in case you make many changes and want to go back to the original version. You can choose ***File>Save As*** and save the file with another name.

Auto Save

This command is found under ***File>Preferences>Filing*** and directs this Designer Elements program to save a backup copy of your work periodically. If your computer hangs up for any reason, the work you did up to the last Auto Save will be recoverable.

You have the option to save after a certain number of commands or minutes. You can also specify the number of backup files created before reusing a backup file name. (See Chapter 6 for more information on setting up or using the Auto Save feature.) For Windows, files opened with the READ ONLY attribute are also autosaved.

Files saved using this feature are placed in the Backup folder within the Designer Elements program folder.

If *Auto Save* is set to on in Preferences but you have not modified your geometry since the last autosave, the function does not activate.

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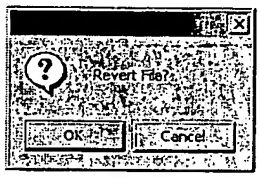
Revert

This command in the File menu deletes all changes made in the drawing since the file was last saved.



1. Choose ***File>Revert***. The following dialog box appears.
2. Click OK to return to the original file opened and close the dialog box. (Click Cancel to close the dialog box without enabling the command.)

Close - CTRL+F4 (Windows);z +F4 (Macintosh)

This command in the File menu closes the current Designer Elements program document. If other Designer Elements program documents are open, they remain open when you close the current document. If you have made any changes since you last saved the current document, a dialog box allows you to save the changes. You can close the document with or without saving the changes.

For Windows, you can also close the document by double-clicking the Control menu at the upper left corner of the title bar. For Macintosh, click the Close button in the upper left corner.

Exit - CTRL+Q (Windows); Quit - z +Q (Macintosh)

This command in the File menu closes this Designer Elements program. If you have made changes since you last saved, a dialog box allows you to save the changes. If you have more than one document open, an alert message allows you to save unsaved documents.

Files and Preferences

In the *Filing* page of the Preferences dialog box you can choose settings for your this Designer Elements program files. These include options for saving native picture formats, clearing undo, compacting files, read-only network file sharing, recent files and auto save.

Files and Locking

Both Windows and Macintosh operating systems provide you with the ability to lock files. Once locked these files can be opened and printed but not altered until unlocked. This feature is valuable when working on a network. When one person is working on the file, another person can open the file but is unable to make changes. See the User Guide for your system for more information.

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Tip:

If your system crashes with an open file, the file will lock. Unlock the file according to your system instructions. You can also save the file under the same name. As you save you will be prompted to confirm the overwriting of the locked file.

File Size & Deleting

When you delete objects in this Designer Elements program, the object data is still retained until a second delete operation is performed. If your notice that your file size is large in comparison to the amount of geometry in the file, this may be the reason. Perform another delete operation (create a line and delete it), save the file and reopen it.

Uninstalling This Program on Windows

When you uninstall this Designer Elements program, the *Prefs.ini* file does not automatically uninstall since it was created after installation. This file is located within the Designer Elements program folder in the Environ folder.

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Importing & Exporting

There are various reasons to use the import and export functions. You might want to bring a document in from another application to work on in this Designer Elements program. You might need to save a document in a format other than the this Designer Elements program format for use with another application. This Designer Elements program offers you a wide variety of options to accomplish this.

The following topics are covered in this chapter:

- Importing
- Exporting

Importing

This Designer Elements program allows you to import many different kinds of files. Some types include import options specific to the format.

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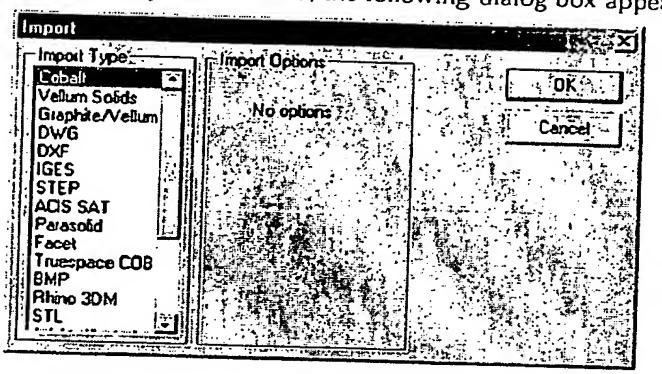
When you choose the *Import* command, the following dialog box appears.



The dialog box contains these options:

Import Type	Lists the file formats that this Designer Elements program imports.
Import Options	Includes the options that are available for the selected format.
OK	Click this button to close the import dialog box and display the standard Open dialog box for locating the file.
Cancel	Click this button to close the dialog box and ends the operation.

Supported Import Formats

This Designer Elements program imports these file formats: Cobalt, Vellum Solids, Graphite/Vellum 3D, DWG, DXF, IGES, STEP, ACIS SAT, Parasolid (Windows only), Facet, Truespace COB, BMP, Rhino 3DM, STL, Adobe Illustrator, 3D Studio, Catia v4, Spline, Text, Grid Surface, ProE/Granite (Windows only) and PICT (Macintosh only).

Vellum Solids

Selecting this type imports files created in Vellum Solids. There are no options for this import type. In order to read previous versions of Vellum Solids into this Designer Elements program, you must import the file using this format.

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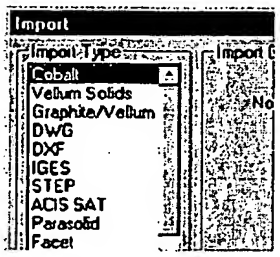
Graphite/Vellum



Selecting this type imports files created in Graphite/Vellum 3D. Graphite/Vellum 3D includes three options: *Vellum Layers*, *Auto Heal Bodies* and *Feature Recognition*.

This translator imports horizontal, vertical, diameter and radial dimensions in addition to geometry.

Vellum Layers

This section includes check boxes, *Create*, *Create Empty* and *Display All*.

Create - When checked, This Designer Elements program creates/imports the layers in the Vellum 3D file and places geometry on their respective layers. If this box is not checked, all geometry is placed on the work layer.

Create Empty - When checked, this Designer Elements program creates layers that contain no geometry. If this box is not checked, layers that contain no geometry are deleted.

Display All - When checked, this Designer Elements program displays all objects on all layers. If this box is not checked, the import file determines which layers display.

Auto Heal Bodies

When checked, this Designer Elements program applies auto-healing to imported surfaces. Auto-healing attempts to find collections of surfaces that define closed volumes and convert them to solids. With this box checked, the *Feature Recognition* check box is enabled.

If this box is not checked, auto-healing is not be performed and the *Feature Recognition* check box is disabled.

Graphite/Vellum 3D Notations

- The Vellum 3D import translator does not support the following entities: work plane, fill, bitmap, balloon, hatch, sheet and view.

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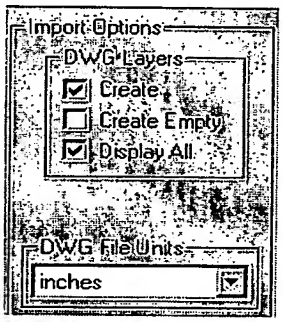
- ACIS does not support skewed ellipses. Only ellipses created with the 2-Point Center Ellipse tool and the Opposite-Corner Ellipse tool in Vellum 3D can be imported in this Designer Elements program.
- This Designer Elements program does not support models (model space). When you import a file with geometry in different models, each model is placed on its own layer.
- When you import a file into this Designer Elements program and you want to surface them to intersect with other objects or project curves onto them, use the Cover, Skin or Net Surface tools.



DWG

Selecting this type imports DWG files created in AutoCAD or other programs that support the DWG file format. The translator will read DWG files up to and including version R2000.

This translator imports horizontal, vertical, diameter and radial dimensions in addition to geometry.

DWG includes three options: *DWG Layers*, *Feature Recognition* and *DWG Units*.

DWG Layers

This section includes check boxes, *Create*, *Create Empty* and *Display All*.

Create - When checked, this Designer Elements program creates/imports the layers in the DWG file and places geometry on their respective layers. If this box is not checked, all geometry are placed on the work layer.

Create Empty - When checked, this Designer Elements program creates layers that contain no geometry. If this box is not checked, layers that contain no geometry are deleted.

Display All - When checked, this Designer Elements program displays all objects on all layers. If

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left unchecked, the import file determines which layers display.

DWG File Units — This list sets the units for the incoming file. The units include inches, feet, mm, cm and meters.

DWG Notations

The DXF/DWG translator does not support the following entities: Shape, Ole2frame, MLine, Leader, MText, ViewPort, Tolerance, Proxy, Hatch and Image.

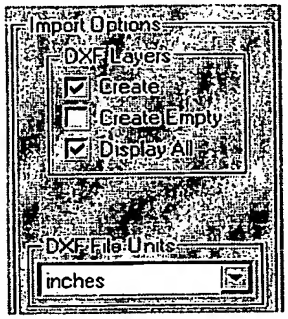
If you import a DWG file that contains entities that are not supported, a warning box will appear telling you that the entities were not read.

See Appendix C for supported DWG entities.

DXF

Selecting this type imports DXF (AutoCAD's Data eXchange Format) files. The translator will read DXF files up to and including version R2000.

This translator imports horizontal, vertical, diameter and radial dimensions in addition to geometry.



DXF includes three options: *DXF Layers*, *Feature Recognition* and *DXF Units*.

DXF Layers — This section includes check boxes, *Create*, *Create Empty* and *Display All*.

Create - When checked, this Designer Elements program creates/imports the layers in the DXF file and place geometry on their respective layers. If this box is not checked, all geometry is placed on the work layer.

Create Empty - When checked, this Designer Elements program creates layers that contain no geometry. If this box is not checked, layers that contain no geometry are deleted.

Display All - When checked, this Designer Elements program displays all objects on all layers. If

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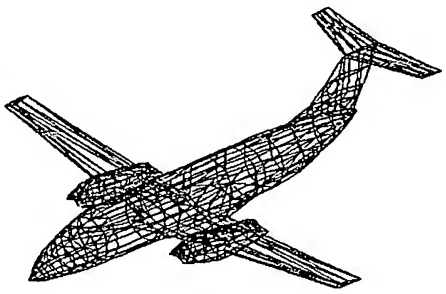
this box is not checked, the import file determines which layers display.

DXF File Units

This pull-down list sets the units for the incoming file. The units include inches, feet, mm, cm and meters.



DXF Facet Files

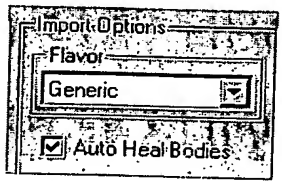
In DXF and DWG files created prior to AutoCAD Release 13, surfaces were represented as a collection of facets. This Designer Elements program converts these surfaces to 3-Point mesh elements. This graphic shows an example of a DXF facet file that was imported into this Designer Elements program.

DXF Notations

The DXF translator does not support the following entities: Shape, Ole2frame, MLine, Leader, MText, ViewPort, Tolerance, Proxy, Hatch and Image.

If you import a DXF file that contains entities that are not supported, a warning box will appear telling you that the entities were not read.

See Appendix C for supported DXF entities.



IGES

Selecting this type imports IGES files created by various CAD programs. IGES includes three options: *Flavor, Auto Heal Bodies* and *Feature Recognition.*

This translator also supports importing horizontal, vertical, diameter and radial dimensions.

Flavor

This section includes a pull-down menu allowing you to choose to import three different IGES flavors: *Generic, AutoCAD* or *SolidWorks.* Use *Generic* if your specific IGES flavor is not listed.

Auto Heal Bodies

When checked, this Designer Elements program box applies auto-healing to imported surfaces.

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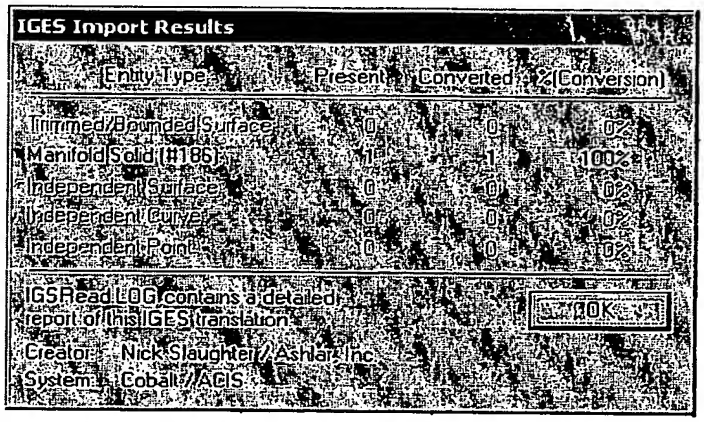
Auto-healing attempts to find collections of surfaces that define closed volumes and convert them to solids. With this box checked, the *Feature Recognition* check box is enabled.

If this box is not checked, auto-healing is not performed and the *Feature Recognition* check box is disabled.

See Appendix D for supported IGES entities.

When you import an IGES file, an IGES Import box appears with the result of your import.



The box contains the following items:

Entity Type — This column lists five entity types: Trimmed/Bounded Surface, Manifold Solid (#186), Independent Surface, Independent Curve and Independent Point.

Present — This section displays the number of a particular entity in the file.

Converted — This section displays the number of entities for the type that converted.

%(Conversion) — This column displays the percentage conversion of the particular entity.

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Creator Information This section contains the name of the person who created the IGES file and system information.

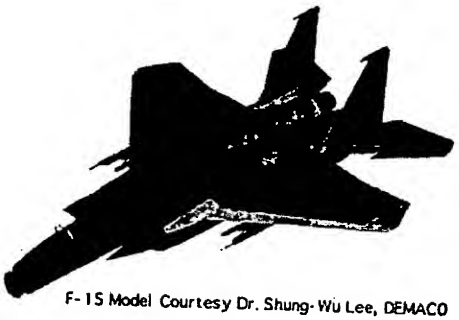
ACIS SAT

Selecting this type imports SAT files created by various ACIS-based CAD programs. There are no options for this type.

Facet

Selecting this type imports ASCII Facet files created by various CAD programs. There are no options for this import type.

The facet file format was created by the United States Electromagnetic Code Consortium and Lockheed Fort Worth. This file format provides an efficient means for transferring geometry models to a variety of government-sponsored signature predication codes. The format includes geometry definitions of 3- and 4-sided facets and material IDs for each facet.



F-15 Model Courtesy Dr. Shung-Wu Lee, DEMACO

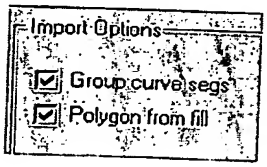
The graphic shows a facet file after it was imported into this Designer Elements program. Each shade represents a material to be analyzed by the signature predication code. The canopy and raydome are transparent, so they are not included in the signature model.

PICT (Macintosh only)

Selecting this type imports Pict files created by various programs. Pict is the Macintosh native file format. There are no options for this import type.

Adobe Illustrator

Selecting this type imports Adobe Illustrator version 5.0 through 8.0 files. Adobe Illustrator includes two options: *Group curve segs* and *Polygon from fill.*



Group curve segs This option sets how curves are imported. Illustrator creates curve seg-

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ments. When selected, separate segments are grouped into one curve. When this option is not selected, curves, like a circle are imported into this Designer Elements program as four arcs.

Polygon from fill — This option sets whether fill is imported with curves if the objects contain fill. When selected, the fill and curves are imported. When this option is not selected, fill is not imported.

This translator also supports importing text.

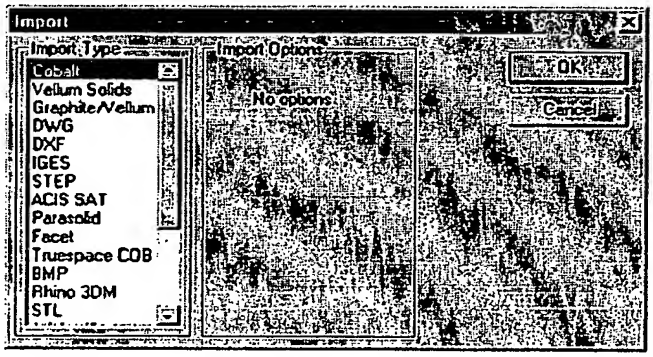
Spline

Selecting this type imports ASCII Spline files created by various CAD programs. There are no options for this import type. See Appendix F for information on how to create spline file for import using a text editor.

Import Command

This command in the File menu imports a document and places it in the current file.

1. Choose ***File>Import***.



   The Import dialog box appears with all the import options.

2. Select the Import type from the list and the options for the type as desired.

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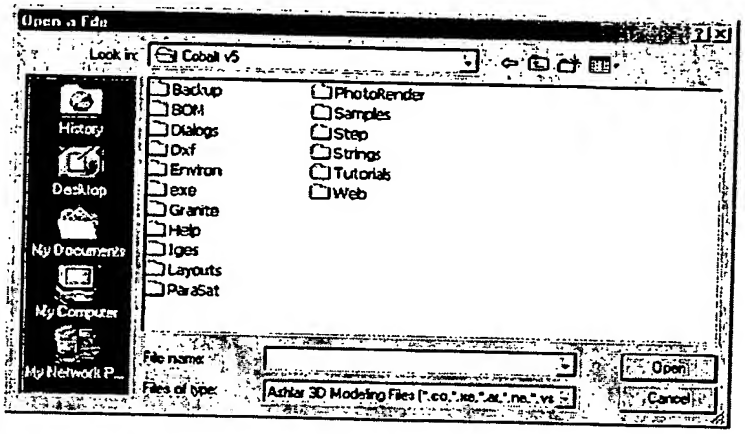
3. Click OK. The standard Open dialog box appears.



4. Navigate to the file you want to import.

   (Windows users: If the *Open as read only* box is checked, the original file cannot be edited but the imported geometry can be.)

5. Click Open to import the file.

The file appears in the drawing area. If you want to save the file in its original format after editing, you must choose the *Export* command from the File menu.

Importing Splines

When you import a text file that contains the coordinates of a spline, Vellum will create the spline according to the imported coordinates.

1. Select ***File>Import***.

   The Import dialog box appears.

2. Specify the import option Spline.
3. Click OK. This Designer Elements program displays the standard Open dialog box.
4. Select a text file that contains the coordinates for the spline.
5. Click Open to import the file.

   This Designer Elements program begins creating the spline.

See Appendix F for information on how to create a text file for importing a spline.

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Importing Tips and Notations

These tips and notations will help you successfully import files.

- **Groups** - This Designer Elements program can import grouped geometry and groups within groups.
- **Layers and Color** - Be careful importing geometry into a layer with a color override. Objects placed on that layer will be displayed in the color of the layer.
- **ACIS Data** - Surfaces and solids data is written out in DXF only.
- **Smart Walls** - This Designer Elements program does not support the Smart Walls feature of Graphite and prior versions of Vellum 3D. Smart walls imported into this Designer Elements program are converted into individual lines without attributes such as height and thickness.
- If this Designer Elements program displays geometry that was not in the original AutoCAD DXF file, go back to the original file in AutoCAD. Choose the PURGE command and purge any unnecessary blocks in the file. Then export the DXF file and import it into this Designer Elements program.
- If an error occurs when you try to import a file, this Designer Elements program creates a file log in the Designer Elements program folder. For example, if you try to import a file through the IGES translator and an error occurs, the file IGSRead is created.
- Importing an IGES, Vellum 3D, SAT, DXF or DWG file with the *Feature Recognition* checked - There may be times when the Feature recognition operation returns a successful code but no geometry displays. This gives the impression that no geometry was imported when in fact *Feature Recognition* failed and discarded the geometry. Import the file with *Feature Recognition* off and the geometry will successfully import.

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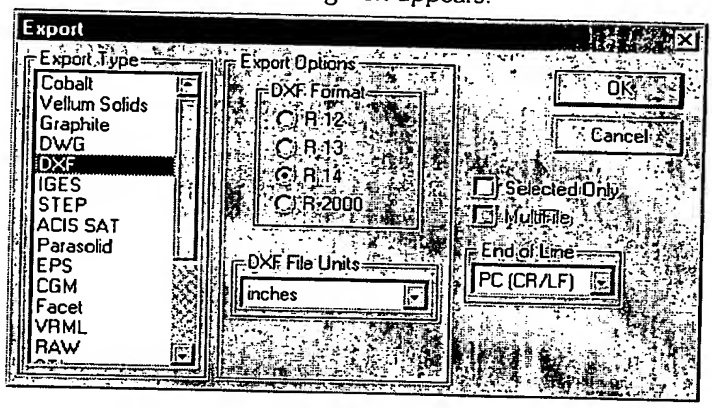
Exporting

This Designer Elements program allows you to export to many different kinds of files formats. Some types include export options specific to the format. When you choose the *Export* command, the dialog box appears.



The dialog box contains these elements:

Export Type	Lists the file formats that this Designer Elements program exports.
Export Options	Includes the options that are available for the selected format.
Selected Only	With this option check marked, only selected geometry is exported.
MultiFile	This option is useful when exporting files to products that do not have the ability to handle multiple objects within one file such as Pro/E and SolidWorks. This option will create a file for each object in your drawing, its name will be the same as in the Design Explorer.
End of Line	This menu allows you to choose an end of line structure for your exported file. You have three options: Mac (LF), PC (CR/LF) and Unix (CR).

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OK	Click this button to close the Export dialog box and displays the standard Save as dialog box.
Cancel	Click this button to close the dialog box and end the operation.

Supported Export Formats

This Designer Elements program exports these file formats: Cobalt,Vellum Solids, Graphite, DWG, DXF, IGES, STEP, ACIS SAT, Parasolid, EPS, CGM, Facet, VRML, RAW, STL, Adobe Illustrator, Catia v4, Text, ProE/Granite, Macrmedia, Viewpoint Media and PICT (Macintosh only).

Vellum Solids

Selecting this type exports files as a Vellum Solids file. You have four options: VS 2000, VS 2000 SP1, VS 99 SP1, VS99 and VS 98.

VS 2000	This format exports Vellum Solids 2000 files. Use this format when exporting from a Vellum Solids 2000 service pack to be read into Vellum Solids 2000.
VS2000 SP1	This format exports Vellum Solids 2000 Service Pack 1 files.
VS 98	This exports the Vellum Solids 2000 file as a Vellum Solids 98 file.
VS 99	This format exports Vellum Solids 99 files.
VS 99 SP1	This format exports Vellum Solids 99 Service Pack 1 files.

Graphite/Vellum 3D

Selecting this type exports files as a Vellum 3D file. This translator exports text and horizontal, vertical, diameter and radial dimensions in addition to geometry. Angle center mark, ordinate and balloon dimensions are exploded into lines and text. There are no options for this type.

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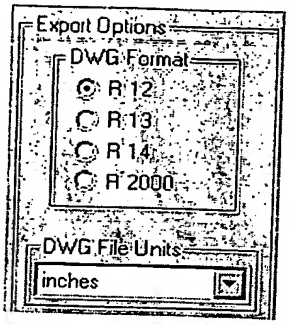
DWG

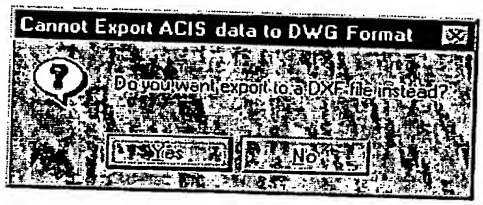


Selecting this type exports DWG files compatible with AutoCAD and other programs that support the DWG file format. (DWG is the binary version of DXF.) DWG includes four format options, *Version 12*, *Version 13*, *Version 14* and *Ver. 2000*.

This translator exports all dimension types: horizontal, vertical, diameter, radial, ordinate, angled, center mark and balloon dimensions. It also exports groups and bezier and vector splines. Hatching is converted into lines.

Warning: Layer names are limited to the following character set: 'a' through 'z', 'A' through 'Z', '0' through '9', '-' and '_'. All other characters in a layer name convert into an underscore (_).

ACIS data cannot be exported using the DWG translator. If you attempt to use this translator on a file containing ACIS data, a warning dialog box appears asking if you want to export the file using the DXF translator.



Click Yes to create a DXF file or No to end the operation.

Version 12 — This option exports the file as an R12 file. This does not support ACIS data. Ellipses, conics, splines are converted into polylines. ACIS curves are converted into b-splines. Surfaces and solids are converted into facets (Face3D).

Version 13 — This option exports the file as an R13 file. Ellipses, splines and ACIS curves are supported. Conics are converted into polylines.

Version 14 — This option exports the file as an R14 file. Ellipses, splines and ACIS curves are supported. Conics are converted into polylines.

Version 2000 — This option exports the file as an R2000 file.

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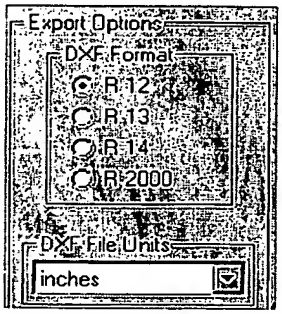
Choose the DWG option based on the translator version supported by the program into which you want to import your file.

DXF



Selecting this type exports DXF files compatible with AutoCAD. DXF includes four format options: *Version 12, Version 13, Version 14* and *Ver. 2000.*

Choose the end of line structure (Mac (LF), PC (CR/LF) or Unix (CR)) from the End of Line pull-down menu.

This translator exports all dimension types: horizontal, vertical, diameter, radial, ordinate, angled, center mark and balloon dimensions. It also exports groups and bezier and vector splines. Hatching is converted into lines.

Warning: Layer names are limited to the following character set: 'a' through 'z', 'A' through 'Z', '0' through '9', '-' and '_'. All other characters in a layer name convert into an underscore (_).

Version 12

This option exports the file as an R12 file. This does not support ACIS data. Ellipses, conics, splines are converted into polylines. ACIS curves are converted into b-splines. Surfaces and solids are converted into facets (Face3D).

Version 13

This option exports the file as an R13 file. Ellipses, splines and ACIS curves are supported. Conics are converted into polylines.

ACIS data for surfaces and solids are written out as SAT data (Spatial Technologies format). A program that supports these versions does not automatically support SAT data. Check your program manual or with the manufacturer to determine whether it can read SAT data. Geometry exported using these versions is considered more accurate than facet representation.

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Version 14

This option exports the file as an R14 file. Ellipses, splines and ACIS curves are supported. Conics are converted into polylines.

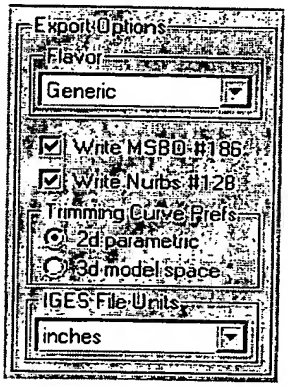
ACIS data for surfaces and solids are written out as SAT data (Spatial Technologies format). A program that supports these versions does not automatically support SAT data. Check your program manual or with the manufacturer to determine whether it can read SAT data. Geometry exported using these versions is considered more accurate than facet representation.

Version 2000

This option exports the file as an R2000 file.

Choose the DXF option based on the translator version supported by the program into which you want to import your file and the end of line setting.

IGES



Selecting this type, exports various versions of IGES files. IGES includes four format options: *Flavor, Write MSBO #186, Write Nurbs #128* and *Trimming Curve Prefs.*

Choose the end of line structure (Mac (LF), PC (CR/LF) or Unix (CR)) from the End of Line pull-down menu.

This format also exports groups.

Flavor

This section includes a pull-down menu allowing you to export different IGES flavors: *Generic, AutoCAD, SolidWorks, Vellum v3.0, Vellum v2.7, Pro/E* or *Alias.*

AutoCAD, R13: Certain MSBOs are not supported by AutoCAD. This flavor converts the MSBOs so they can be read in AutoCAD.

SolidWorks: This flavor does not support IGES Conic Arc (#104) which this Designer Elements

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program uses to write an ellipse. Ellipses convert into nurb splines.

Vellum v3.0: All solids convert into surfaces.

Vellum v2.7: All solids and surfaces convert into curves.

Pro/E: Does not include the Trimming Curve Prefs options.

Alias: Includes all the options listed for the Generic flavor.

Use Generic if your specific IGES flavor is not listed.

Write MSBO #186 — Check this box to export solids using this IGES5 solid object type. (MSBO #186 is a Manifold Solid B-Rep entity.) If this box is not checked, only 3D parametric trimming curves for analytic surfaces are exported.

Write Nurbs #128 — Check this box to export solids using this nurb surface type.

Trimming Curve Prefs — This section includes two check boxes *2D Parametric* and *3D Model Space*. Choose one option.

2D Parametric - Check this option to export 2D parametric trimming curves for analytic surfaces. This option is valuable to programs that can read only 2D data and need the 3D data mapped to a 2D parametric.

3D Model Space - Check this option to export the actual 3D trim curve in the model space.

See Appendix D for the supported IGES entities.

STEP

Selecting this type exports a STEP file. STEP (STandard for the Exchange of Product model data) is a neutral file format used to export models among CAD, CAM and CAE applications.

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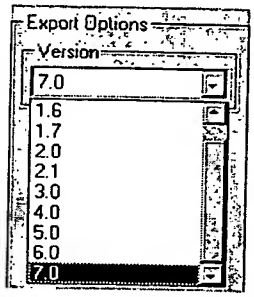
You can also choose the end of line structure (Mac (LF), PC (CR/LF) or Unix (CR)) for your file from the End of Line pull-down menu. There are no other options for this export type.

ACIS SAT

Selecting this type exports SAT files compatible with various versions of ACIS. This format includes one option: *Version.*



Version — The option sets the ACIS export version. The pull-down menu includes: 1.5, 1.6, 1.7, 2.0, 2.1, 3.0, 4.0, 5.0, 6.0 and 7.0.

This translator supports exporting curves, surfaces, solids and grouped objects. It does not support exporting layers.

Tip:

This Designer Elements program EPS exports an embedded post script file. This type is great for printing and embedding in such applications as Microsoft Word and PowerPoint.

This EPS export is different from the EPS export used in Vellum 3D which exports an Adobe Illustrator version of post script.

EPS

Selecting this type exports an EPS (encapsulated postscript) file. There are no options for this type.

CGM

Selecting this type exports a computer graphics metafile. This format is a 2D data exchange format which allows graphical data to be stored and exchanged among graphics devices, applications and computer systems. This metafile is not a picture but a description of a picture. There are no options for this translator.

Facet

Selecting this type exports an ASCII Facet file. There are no options for this type.

STL

The precise, mathematical representation of a solid or surface must often be converted into a collection of imprecise planar facets. These facets may be used to export a model to the STL format and when changing a solid or surface

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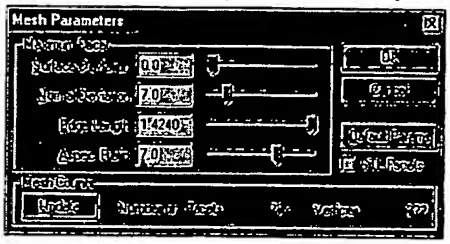
(Edit>Change Object Type...) to a mesh. The amount of error that results from this conversion is controlled by the settings in the mesh parameters dialog box.



During the conversion, vertex points are distributed on the surface or solid. These vertices are then grouped into 3-sided and 4-sided facets. The conversion is deemed acceptable when the generated vertices and facets satisfy the settings. The 5 available settings are Surface Deviation, Normal Deviation, Edge Length, Aspect Ratio, and STL Facets. These settings are defined in the sections below.

Change the facet settings as needed in the dialog and then click the Update button to see the number of facets and vertices generated. Determining the combination of settings that will work for a given situation can be a little bit of an art. If one setting becomes too tight, the other settings will have no effect. If one setting becomes too loose, it will have no effect.

Keep in mind that the settings are used by the faceting algorithms if possible. It is often not possible to satisfy all settings simultaneously. In this situation, the algorithm decides which settings to "loosen".

The Mesh Parameters dialog box contains the following options:

Surface Deviation This setting controls the maximum allowed distance between any point on the actual surface or solid and the facet representing that point. The exaggerated figure below shows the largest dis-

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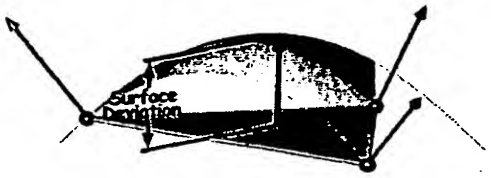
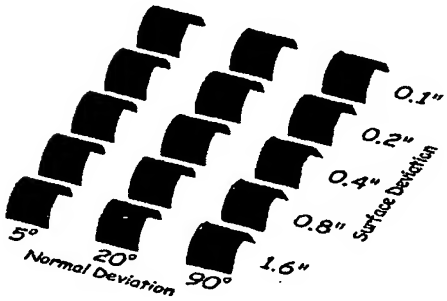
tance between a patch on the actual surface (yellow) and the corresponding planar facet (brown).



Normal Deviation This setting controls the maximum allowed angular difference between any normal on the actual surface or solid and the corresponding interpolated normal on the facet.



Edge Length This setting controls the maximum allowed edge length of any given facet.

Aspect Ratio This setting controls the maximum allowed aspect ratio of any given facet.

STL Facets This setting will force the facets generated to be suitable for stereolithography usage. This setting is usually used when exporting STL files.

VRML

Selecting this type exports virtual reality modeling language files. There are two options for VRML, Version 1.0 and Version 2.0.

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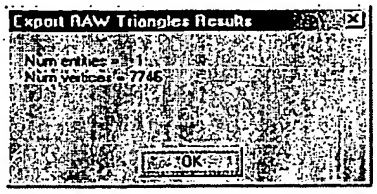
RAW

Selecting this type exports a raw file consisting of triangular vertices. These vertices define the x, y and z locations of the 3D faces which make up your model. There are no options for this type.



After exporting, a dialog box appears displaying the number of entities and vertices in the exported model.

You can view the raw file by opening it in any text editor.

Adobe Illustrator

Selecting this type exports an Illustrator file. This Illustrator file is compatible with Adobe Illustrator versions 5.0 through 8.0. This translator supports exporting the Hidden and Hidden w/dimmed render options. All dimensions are converted into lines and text. There are no options for this type.

Macintosh only: When you export an Illustrator file, this Designer Elements program automatically displays it with an Illustrator icon. If you double-click the file, Adobe Illustrator launches.

Text

Selecting this option will export the text contained in the file along with a list of objects contained in the file.

Catia v4

Selecting this type exports a CATIA v4 file. You can choose to export only the selected items and have each item export into its own file with the MultiFile option checked.

ProE/Granite

Selecting this type exports a file that is based upon the Granite kernal that PTC uses in the Pro/E program. This provide a kernal to kernal export for more accurate translations. There are five options for this export option: Granite, Neutral, IGES, Step and VDA.

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Macromedia

Selecting this type exports a Macromedia file. You can choose the Geometry Quality and you have the option of creation a log file if you check the box.

ViewPoint Media

Selecting this export type exports a Viewpoint Media file with an .mts or .mtx file extension. The .mtx file contains the xml representations of each object as a mesh object, the .mts file contains a list of all objects that should be displayed along with their display status. For futher information please refer to the Viewpoint developers guide or examine the sample .html code generated by our 3D Web Publish command. To view the .mtx/.mts files you will need a viewpoint compatible 3D web creations product such as Adobe Atmosphere.

To skip the export and view this file type immediately choose **File>3D Web Publish**, your geometry will be displayed in your web browser.

PICT (Macintosh only)

Selecting this type exports Pict files, the Macintosh native file format. There are no options for this type.

Export Command

This command in the File menu saves a document in the format you specify.

1. Choose ***File>Export***.

   The Export dialog box appears.

2. Select the export type and its options. If you wish to export only selected objects on the screen, rather than the entire document, click *Selected Only*.
3. Click OK. The standard Save document as dialog box appears prompting you for a name and location for the exported file.

   Windows: The type of file you are exporting is indicated with the appropriate filename extension.

4. Enter the file name and click Save. The exported file is saved with the name and location entered.

Exporting Tips and Notations

These tips and notations will help you successfully export files.

- Windows - this Designer Elements program does not support exporting bitmaps.

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- When exporting files from this Designer Elements program for import into another program, determine what version of the translator is used by the other program. Choose the appropriate translator in this Designer Elements program. Be sure to check whether there are any tips or notations for a successful translation in this Designer Elements program or the other application.
- In general, it is best show all layers before you export from this Designer Elements program or from another program that you will import into this Designer Elements program, so that you know what you are exporting.
- AutoCAD does not like the & symbol in the layers table of the DXF file. Remove the symbol from your layer name in this Designer Elements program or AutoCAD will generate an error reading in the file.
- If line patterns do not import into AutoCAD accurately from a Designer Elements program DXF or DWG file, change the AutoCAD variable LTSCALE to display the patterns at an appropriate scale.

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Drawing Composition

This Designer Elements program allows you to quickly create 2D drawings from one or more surface or solid objects. The 2D drawing views are bi-directionally associative. Changes made to the objects automatically appear in the 2D drawings. Likewise, you can change objects in the 2D drawing by using the Edit Objects dialog box. These also update on your objects. The topics covered in this chapter include:

- Model to Sheet Command
- Drawing Views
- Undfolding Views
- Editing a Drawing View
- Layout Templates
- Printing the Layout

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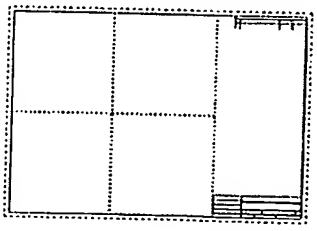
Model to Sheet Command



The Model to Sheet command, located in the Layout menu, automates the process of creating drawings through the use of templates. Templates are empty drawings with pre-defined and pre-arranged drawing views set within drawings borders. With this feature, this Designer Elements program creates 2D geometry from the 3D surface and solid models you selected and places them in drawing views on a new layer, called Sheet View. Your original 3D geometry remains.

Tip:

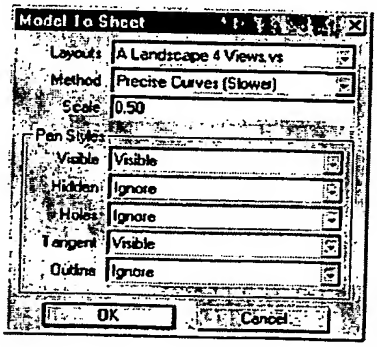
If you turn on all layers and zoom out, your original geometry is visible in the Top view.

Warning: You cannot use this command with curves or groups.

If your geometry contains objects or drawing border lines using the background color, this Designer Elements program automatically changes it to the foreground color. This capability is especially helpful when your background color is black. See Chapter 6 for information on setting the background and foreground colors.

The new sheet view displays in the Top view. Drawing views placed on this sheet can only be selected when the view orientation is Top. The sheet is positioned at the origin using the lower left corner as the alignment reference.

Choosing the Model to Sheet command displays the following dialog box.



The dialog box contains these options:

Layouts

This option sets the layout for displaying your geometry. These layouts are predefined and located in the Layout folder within the same folder as the Designer

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Elements application. The layouts contain drawing views and may contain title blocks and drawing size boundaries depending on the layout selected.

You can create or modify any of the templates based on your drawing layout preferences. See the "Layout Templates" section at the end of this chapter for more information.

Method — This option sets how hidden edges display in the final drawing. You have five settings, Precise Curves (Slower), Polylines (Faster), Flat, Gouraud and Phong.

Tech Note:
The hidden line settings for Edge display do not affect the Hidden Line render mode available in the Render Options dialog box. See Chapter 33 for information on the render modes.

Pen Styles — These option are covered in the Draw View Properties section of this chapter.

Scale — This option sets the scale of the drawing.

OK, Cancel — Clicking OK completes the Model to Sheet command and displays your geometry based on the specified settings. Clicking Cancel, cancels the command and closes the dialog box.

Using the Model to Sheet Command

1. Select the models you want to display on the sheet.
2. Choose ***Layout>Model To Sheet.***

   The Model to Sheet dialog box displays.
3. Choose the Layout and Edge Display from their respective pull-down lists.
4. Enter a scale in the Scale field.
5. Click OK.

   2D geometry is created from your 3D model, placed in the drawing views of the selected layout, and displayed in the Top view. A Sheet View layer is also created on which the drawing views containing your geometry are placed.

   If you selected a layout with a format, A, B, C, D or E, the format is also placed on the Sheet View layer. Choose ***Layout>Layer Manager*** to see the new layer and the num-

Referral:
Drawing views are explained later in this chapter.

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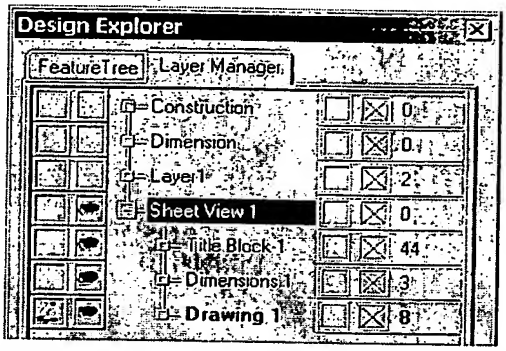
ber of new 2D objects on that layer.



You can also see that the Sheet View layer is now the work layer and all other layers are turned off.

Undoing the Model to Sheet Command

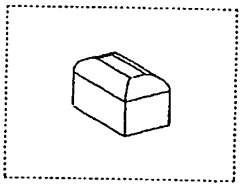
If you inadvertently selected the Model to Sheet command or later want to remove the sheet, you can use the Undo command or the Layer Manager.

- Using Undo: Choose ***Edit>Undo*** (CTRL+Z (Windows) or z+Z (Macintosh). You may have to use the Undo command more than once depending on the number of operations conducted since the Model to Sheet command was chosen.
- Using the Layer Manager: Display the Layer Manager. Choose another layer as the work layer. Then select the Sheet View layer and delete it. Click OK to close the dialog box. Your view is still set to Top. Change your view and zoom scale to view your geometry.

Tech Note:

Drawing views operate differently from those in Vellum 3D. You are not looking at a 3D model in a drawing view but rather 2D wireframe object created from the 3D model. You cannot use the Trackball to rotate the view orientation of a specific drawing view or zoom in the drawing view. However, you can use the *Change View* command and *Properties* command in the Drawing View menu to make adjustments. See the "Drawing View Menu" section later in this chapter.

Drawing Views



(Cobalt™ and Xenon™ Only)A drawing view is a defined area into which the 2D geometry is placed when using the Model to Sheet command. The drawing view displays the geometry in a particular view orientation, like Top, Trimetric, etc. The example here shows a drawing view with a Trimetric view orientation.

Drawing views are created automatically using the Model to Sheet command or by using one of the Sheet tools. Drawing views are placed on top of the Sheet view layer rather than

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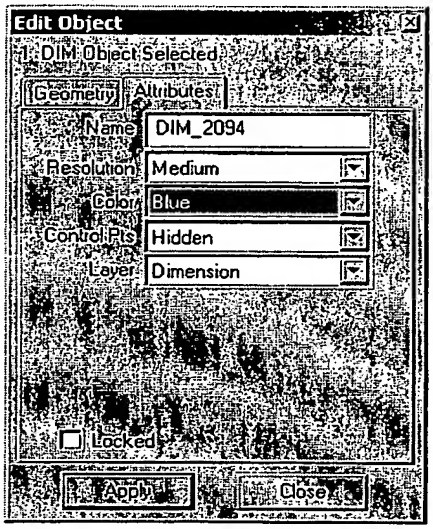
on the layer. Each drawing view has its own work plane. Any additional objects, text or dimensions placed in the view appear only in that view on that work plane, set to Top.

The drawing view frame uses the foreground color set in Preferences. If the Show Triad command is activated, the Triad displays in each drawing view.

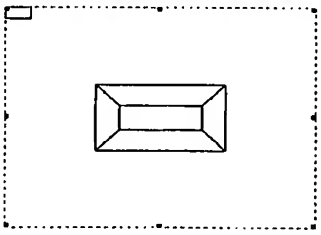
Spacing for crosshatching, dimensions, text, arrow size and line font in drawing views can now be set relative to the sheet through the Edit Objects dialog box, independent of the scale set in the drawing view. This allows you to set any scale for the view without concerning yourself with how the item will appear.

Just double-click the item to display the Edit Objects dialog box. Select the Attributes page and the Ignore Scale option. Click Close or Apply to update the selected item.



Activating Views

Once you've created a drawing, you activate individual views by clicking in the rectangular region surrounding the view. When activated, the view boundaries highlight in red.



You cannot activate drawing views if the sheet is displayed in any view other than Top. All geometry in the view becomes available for selection when the view is activated. The Drafting Assistant recognizes all snap points in the view. Click outside the area to deactivate the view.

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Tech Note:

The Drafting Assistant only recognizes geometry within an active view.

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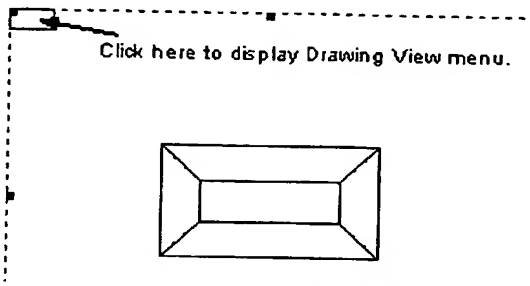
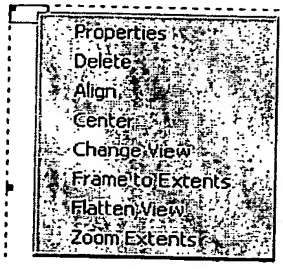
Drawing View Menu

Drawing views have specific commands associated with them. These are accessible through a pull-down menu in the drawing view. Click the mouse in the upper left region of the view.



The view menu displays.

You can also display the view menu by clicking the right mouse button (Windows) or by holding down the CONTROL key and clicking the mouse button (Macintosh).



The view menu contains the following commands: Properties, Delete, Align, Center, Change View, Frame to Extents and Flatten View.

Properties Command

The Properties command gives you access to a set of properties that affect a drawing view's display on the screen. When you choose the command, the following dialog box displays.

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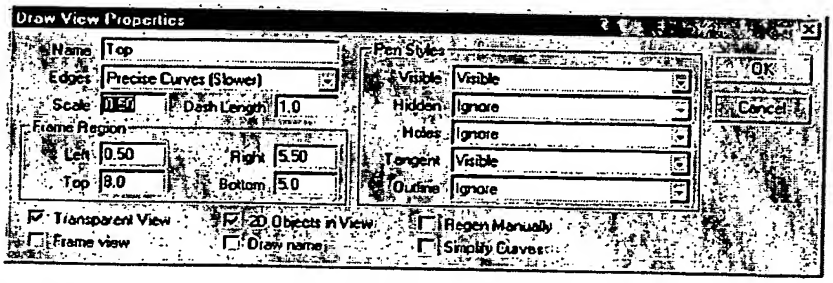

The dialog box contains the following options:

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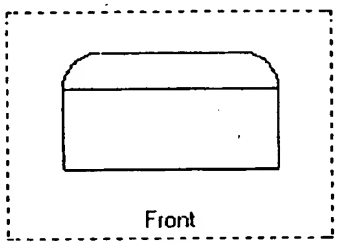
Name

This field allows you to specify the name that appears in the drawing view when you activate the Draw name check box in the dialog box.



Edges

This option sets how hidden edges display in the final drawing. You have five settings, Precise Curves (Slower), Polylines (Faster), Flat, Gouraud and Phong.

Scale

This field sets the scale of the drawing.

Dash Length

This field sets the length of dashes when using either the Dash All Hidden (precise) or Dash Only Holes (precise) edge display options.

Frame Region

This region defines the rectangular area of the selected drawing view and its location. The field values represent the view edge's distance from the origin (where the sheet is automatically placed).

The Left field sets the distance from the left view edge to the origin (0,0,0). The Right field sets the distance from the right view edge to the origin (0,0,0). The Top field sets the distance from the top view edge to the origin (0,0,0). The Bottom field sets the distance from the bottom view edge to the origin (0,0,0).

Units are based on those set in Preferences.

Pen Styles

Pen styles are used to define edge attributes options within a draw view. Edge options include visible, hidden, holes, tangent and outline. A pen style defines the pen weight, pattern and color.

Tech Note:

The hidden line settings for Edge display do not affect the Hidden Line render mode available in the Render Options dialog box. See Chapter 33 for information on the render modes.

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One advantage to using a pen style to define an edge attribute is the ease at which you can then explore different pen styles for all your draw views. Any change to the master pen style will automatically update all edges that use that pen style.

For example, suppose you want your visible lines printed in a E size drawing at a heavier weight than an A size drawing. In this case just change the Visible pen weight to your desired weight and print.

Note: You can also select "Use Object Color" which does not use a pen style but always follows the object color used to create the draw view.

Visible The Visible edge format is applied to edges that are not hidden. However this excludes edges that are classified as tangent or outline.

Hidden The hidden edge format is applied to edges that are occluded.

Holes The holes edge format is applied to edges that are hidden and cylindrical.

Tangent The tangent edge format is applied to edges that are shared by two faces that are tangent. Fillet edges are examples of tangent curves.

Outline An edge is classified as an outline if the normal to the face on one side of the edge points towards the eye and the normal to the face on the other side of edge points away from the eye.

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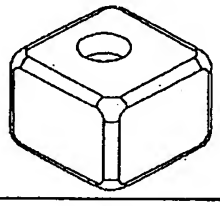
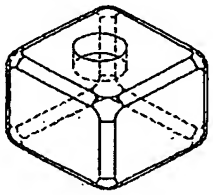
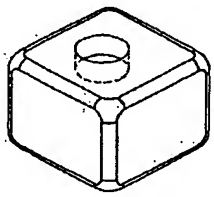
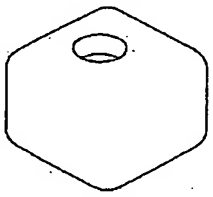
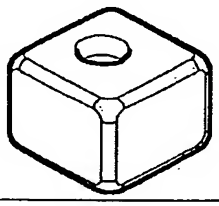
The following picture gives examples of how changing these parameters affect your model.

Visible = Visible Pen Style Hidden = Ignore Holes = Ignore Tangent = Visible Pen Style Outline = Ignore	
Visible = Visible Pen Style Hidden = Hidden Pen Style Holes = Hidden Pen Style Tangent = Visible Pen Style Outline = Ignore	
Visible = Visible Pen Style Hidden = Ignore Holes = Hidden Pen Style Tangent = Visible Pen Style Outline = Ignore	
Visible = Visible Pen Style Hidden = Ignore Holes = Ignore Tangent = Ignore Outline = Ignore	
Visible = Visible Pen Style Hidden = Ignore Holes = Ignore Tangent = Visible Pen Style Outline = My Thick Pen Style	

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Transparent View This check box controls the whether the drawing view background is clear or opaque. When selected, the

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background is clear and objects located underneath the view are visible through the view.

Remember that drawing views are placed on top of the Sheet view layer rather than on the layer. Geometry placed on the layer may fall underneath a view.

Frame view

This check box controls the display of the drawing view frame. When selected, the view frame is visible.

Regen Manually

This check box controls whether the view regenerates or updates automatically after making changes to your geometry. This control is helpful if you have a file that requires a long regeneration time. With this option selected, you can make a series of changes without having to deal with the delay of regeneration. If you want to regenerate one particular drawing view, toggle this option between on and off.

This option relates to the Manual View Regeneration tool in the Sheet tools palette which regenerates all views on the sheet.

2D Objects in View

This check box controls the 3D nature of geometry within the drawing view. When checked, all geometry is flattened to 2D geometry. For example, an arc in a Trimetric drawing view becomes an ellipse in 2D when this option is checked. When left unchecked, no objects are flattened.

Draw name

This check box controls the display of the text in the Name field. When selected, the text displays at the lower border of the frame, centered between the left and right edges.

Simplify Curves

The simplify curve option will attempt to simplify splines into lines, arcs, or circles when appropriate. This is primarily useful if the objects used in the model to sheet consisted of splines instead of the typical analytics.

Auto Hatch

(Only appears for section views) This check box controls hatching for a section view. When selected, hatch

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automatically appears within section geometry. This option uses the default hatch set in the Cross Hatch dialog box. See Chapter 28.

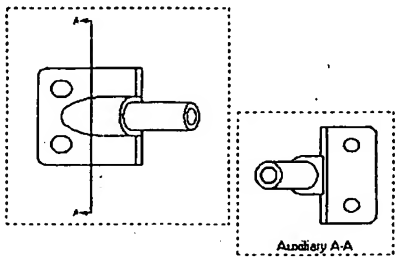
Area Hatch Only — (Only appears for section views) This check box controls the geometry that appears in the section view. When selected, only the geometry cut by the section displays. When not selected, the section view shows the section geometry and any portion of the geometry lying behind the section cut.

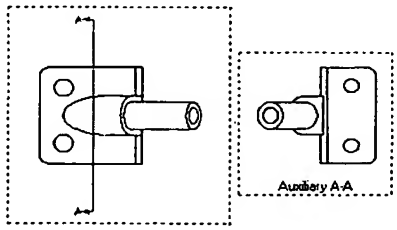
Delete Command

This Delete command removes the drawing view and its contents from the drawing. The original model remains.

Align Command

The Align command functions like a tool in this Designer Elements program in that the Message Line contains steps for its use. This command aligns an active view with another selected view. You can use the command on section views, general drawing views and auxiliary views. The left graphic here shows two views, the Top view of a CAM part with its associated auxiliary view. The right graphic shows the same two views aligned.





1. Select the view you want to align.
2. In the Drawing View menu, choose the Align command.

   The Message Line reads: *Align View: Pick parent view to align with.*
3. Select the parent view.

   The two views align.

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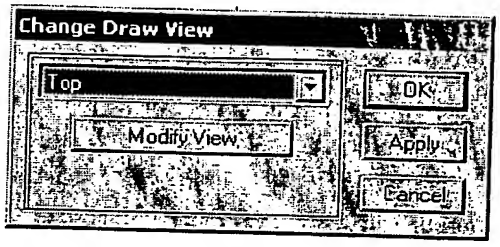
Center Command

The Center command centers the objects within the frame boundaries. This is especially helpful if you've moved the drawing view or changed the view orientation of the geometry.

Change View Command

The Change View command allows you to change the view orientation of the selected drawing view. You can change the view to one of the pre-defined or userdefined views or modify the view by specifying the Eye/Reference point, Azimuth/Elevation or rotation values.

When you choose this command, the Change Aux View dialog box appears.



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The dialog box contains the following options:

View setting

This setting displays the name of the current view and a pull-down list for selecting a different view. The views include Right Side, Left Side, Front, Back, Top, Bottom, Iso (Top Front Left), Iso (Top Front Right), Iso (Top Back Left), Iso (Top, Back Right), Iso (Bottom, Front Left), Iso (Bottom Front Right), Iso (Bottom Back Left), Iso (Bottom Back Right), Trimetric and any user-defined views.

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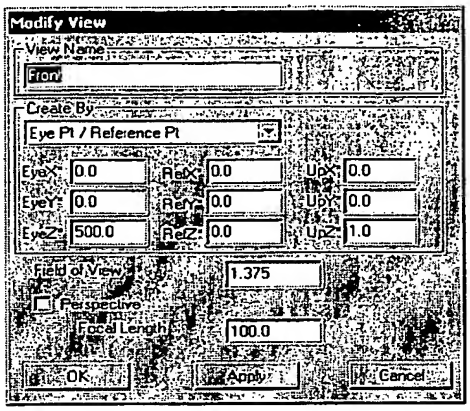
Modify View

Choosing this button brings up the Modify View dialog box.



Change the Eye/Reference point, Azimuth/Elevation or rotation values for the view. You can also name the view. If you enter a name and want to display it in your drawing view, you must also enter the new name in the Draw View Properties dialog box. This view is specific to the drawing view and not available through the Trackball.

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OK

Click this button to close the dialog box and the view changes.

Cancel

Click this button to close the dialog box without making any changes.

Apply

Click this button to change the view but keep the dialog box open.

Frame to Extents Command

The Frame to Extents command shrinks the frame to the size of the geometry in the view.

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Flatten View Command

The Flatten View command deletes the drawing view, projects all objects into 2D entities and places them on the sheet creating complete 2D geometry. This command destroys any associativity between the flattened objects and the 3D model that created them.

The objects are scaled according to the value set in the Properties dialog box for the view. If your view contains dimensions, they are flattened also. The dimension value from the original view is placed in the flattened view. If the view scale was set to anything other than 1.0, the actual measurement of the flattened object will be different than the dimension value. For example, if you dimensioned an object in a view that was 1.25 inches and shown at a scale of 2, the flattened dimension would still read 1.25. However, the actual length is 2.50. I

Important: Be sure to mark your drawings, "Not to Scale," if flattening a view with any scale other than 1 so when people read the dimension value rather than measuring the object.

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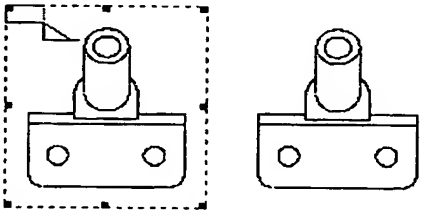
Since the geometry in the view is scaled, its size on the screen after being flattened, is the same as it was in the view. The left graphic here shows a selected view. The right graphic shows the same geometry flattened.



If you flatten a detail view, the detail view boundary is converted into a circle with a phantom pen pattern and flattened onto the sheet with the geometry.

Zoom Extents Command

The Zoom Extents command changes the scale of the drawing view so the objects within it fill the drawing view. This command operates similarly to the Zoom All command except that it applies to a drawing view.

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Sheet Tools

After you've created the drawing, you may want to make some changes to the drawing to better illustrate certain aspects of your model. This Designer Elements program provides you with tools to create new views, modify a selected view and add auxiliary views, section views and details views to your drawing. These tools are available through the Sheet tools palette. The tools include Drawing View, Auxiliary View, Section View, Detail View and Manual View Regeneration. Choose ***Window>Sheet Tools*** to display the palette.

Drawing View Tool

This tool allows you to create empty drawing views. Use this tool to add drawing views to your current layout already containing views or create customized layouts.

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Using the Drawing View Tool

1. Select the Drawing View tool. The Message Line reads: *Drawing View: Pick start position view frame.*
2. Click the starting point position for the frame. The Message Line now reads: *Drawing View: Pick end position view frame.*

   Notice that as you move the pointer to select the ending position an outline of the frame appears.
3. Click the ending point position for the frame.

   The new view appears with the frame edges. The view orientation is Front. Continue placing new views as desired. To activate the view, choose the Selection tool and select the frame

Adding Associative Geometry to Empty Drawing Views

This Designer Elements program allows you to add geometry (associative to the original 3D geometry) to empty views. When changes are made to the geometry all associative geometry updates.

1. Turn on the layers containing the 3D geometry you want to place in the new view.

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2. Select the 3D geometry.

   If you select geometry from an existing drawing view, you would only be copying 2D wireframe geometry.

3. Choose ***Edit>Copy***.
4. Select the empty drawing view.
5. Choose ***Edit>Paste***.

   The geometry pastes into the new view. Use the Drawing View menu to center the geometry, change the view, etc.

Auxiliary View Tool

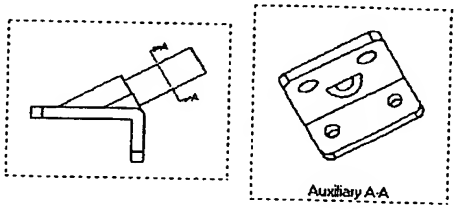
This tool allows you to create auxiliary views from a referenced drawing view. Auxiliary views dynamically align to that view and are associative to both the original view and the auxiliary dimension. Change the geometry and auxiliary view automatically updates. Move the dimension location in the original view and again the auxiliary view reflects the changes. Auxiliary dimension lines appear in the original view. These dimensions are placed on the Sheet View layer. The Auxiliary tool can only be used when a drawing view is present.

Tech Note:

The font type, size or style of the dimension label cannot be changed. Use the Text tool to create to create your own labels.

By default, auxiliary labels are alphabetical. If you place more than one auxiliary view in your drawing, the auxiliary view label automatically increments to the next letter. If you close the file containing auxiliary views and later reopen it and add another auxiliary view, the label increments to the next letter based on the last auxiliary label in the file. You can also enter any label text desired. However, user-defined label text does not automatically increment.

The left graphic here shows the referenced view with auxiliary view dimension lines. The right graphic shows the auxiliary view.



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Using the Auxiliary View Tool

1. Select the Auxiliary View tool. The Message Line reads: *Auxiliary View: Pick drawing view for auxiliary view.*
2. Select the drawing view from which you want to create the auxiliary view.

   The Message Line now reads: *Auxiliary View: Pick start and end of folding line.*
3. Click the start and endpoint locations that specify the folding line for the view. The points do not need to be located on the geometry.

   The Message Line reads: *Drag window to final position.*
4. Drag the view to a new location and click the mouse button. Notice that as you move the pointer an outline of the frame appears.

   The auxiliary view displays with a view label at the specified location. The auxiliary view dimension appears in the original view.

   If you want to delete the view later, you must delete both the view and the dimension line in the original view. You may have to adjust the auxiliary labels to accommodate the deletion.

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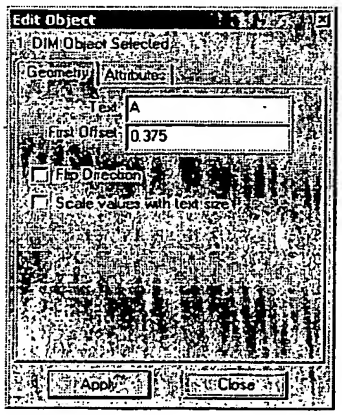
Changing the Direction or Label of the Auxiliary View

If you want to change the auxiliary view direction opposite to that indicated by the view dimension, select the dimension and choose ***Window>Edit Objects***. In the Geometry page, check the Flip Direction option and click Apply. The direction of the view changes.

You can also change the view label in this dialog box.



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Section View Tools

These tools create horizontal, vertical and 2 point section views. Section view geometry is associative to the geometry in the view and the section dimension. Change the geometry and the section view automatically updates. Move the dimension location in the original view and again the section view reflects the changes. Section view dimensions are placed on the Sheet View layer.

The Section tools allow you to specify whether you only want to see the section cut or both the section cut and the geometry lying behind it. This ability is controlled in the Draw View Properties dialog box, accessed through the Drawing View menu. See the "Drawing View Menu" section earlier in this chapter.

Tech Note:

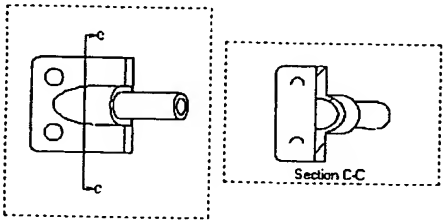
The font type, size or style of the dimension label cannot be changed. Use the Text tool to create to create your own labels.

By default, section labels are alphabetical. If you place more than one section view in your drawing, the section view label automatically increments to the next letter. If you close the file containing section views and later reopen it and add another section view, the label increments to the next letter, based on the last section label in the file. You can also enter any label text desired. However, user-defined label text does not automatically increment.

Tech Note:

You cannot paste solids into section views. If you want to add a solid, you must add it to the original view.

Section views also support crosshatching. These tools can only be used when a drawing view is present. The graphics here show a vertical section view.



When you select the Section View tool, a subpalette appears containing three tools: Vertical, Horizontal and 2 Pt.

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Vertical Section View Tool

This tool creates a vertical section view.

Using the Vertical Section View Tool

1. Select the Section View tool.
2. Select the Vertical Section View tool in the Message Line. The Message Line reads: *Section View: Pick drawing view for section.*

   If a view is already selected, skip to step 3.
3. Select the drawing view.

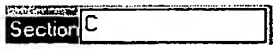
   The Message Line now reads: *Section View: Pick location for vertical section.*
4. Click a point on your geometry for the section.

   The Message Line reads: *Drag window to final position.*
5. Drag the view to a new location and click the mouse button. Notice that as you move the pointer an outline of the frame appears.

   The section view displays with the view label shown in the Status Line. If you want a different label, enter it in the Section data field and press ENTER (Windows) or RETURN (Macintosh). The section view dimension appears in the original view.

   If you want to delete the view later, you must delete both the view and the dimension line in the original view. You may have to adjust the section labels to accommodate the deletion.

The Status Line contains the section view label.



Horizontal Section View Tool

This tool creates a horizontal section view.

Using the Horizontal Section View Tool

1. Select the Section View tool.
2. Select the Horizontal Section View tool in the Message Line. The Message Line reads: *Section View: Pick drawing view for section.*

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If a view is already selected, skip to step 3.

3. Select the drawing view.

   The Message Line now reads: *Section View: Pick location for horizontal section.*

4. Click a point on your geometry for the section.

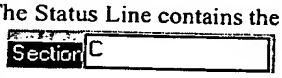
   The Message Line reads: *Drag window to final position.*

5. Drag the view to a new location and click the mouse button. Notice that as you move the pointer an outline of the frame appears.

   The section view displays with the view label shown in the Status Line. If you want a different label, enter it in the Section data field and press ENTER (Windows) or RETURN (Macintosh). The section view dimension appears in the original view.

   If you want to delete the view later, you must delete both the view and the dimension line in the original view. You may have to adjust the section labels to accommodate the deletion.

The Status Line contains the Section view label.



2 Pt Section View Tool

This tool creates a section view based on the orientation of two user-defined points.

Using the 2 Pt Section View Tool

1. Select the Section View tool.

1. Select the 2 Pt Section View tool. The Message Line reads: *Section View: Pick drawing view for section.*

   If a view is already selected, skip to step 3.

2. Select the drawing view.

   The Message Line now reads: *Section View: Pick start and end of section orientation.*

3. Click two points on your geometry to indicate start and end points for the section cutting line.

   The Message Line reads: *Drag window to final position.*

4. Drag the view to a new location and click the mouse button. Notice that as you move the pointer an outline of the frame appears.

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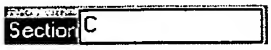
The section view displays with the view label shown in the Status Line. If you want a different label, enter it in the Section data field and press ENTER (Windows) or RETURN (Macintosh). The section view dimension appears in the original view. You may have to adjust the section labels to accommodate the deletion.

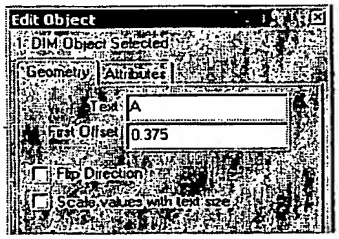
If you want to delete the view later, you must delete both the view and the dimension line in the original view.

The Status Line contains the Section view label.



Changing the Direction or Label of a Section View

If you want to change the section view direction opposite to that indicated by the view dimension, select the dimension and choose ***Window>Edit Objects***. In the Geometry page, check the Flip Direction option and click Apply. The direction of the view changes.



You can also change the view label in the dialog box.

Sections and Crosshatching

By default, section views automatically contain crosshatching. You can remove the crosshatching or change the hatch pattern for a selected view through the Cross Hatch dialog box.

You can also set the default hatch pattern through this dialog box. When nothing is selected, choose ***Pen>Cross Hatch***. See Chapter 28 for more information on the dialog box and the hatches available.

If you have more than one object cut in the section, you can define separate hatch patterns.

Tech Note:

The crosshatching used for geometry in a section view must be set separately from the material set in the Mass Properties dialog box. See Chapter 24 for more information on Mass Properties.

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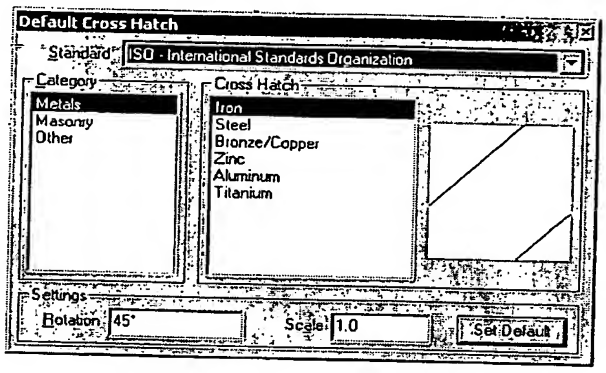
1. In the section view, select the hatch pattern.
2. Choose ***Pen>Cross Hatch***. The following dialog box displays.
3. Select a category and Cross Hatch pattern from their respective lists.
4. Set the hatch rotation angle and scale.
5. Click Apply to change the selected hatch.

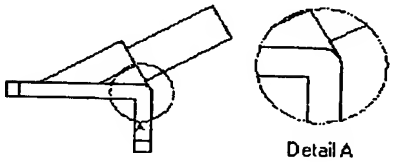


If you are not satisfied with the hatch pattern, choose another. You can also change the hatch patterns of any other section geometry in the view or in other views.

6. Click the Close button (Windows) or the Close box (Macintosh) to exit the dialog box when you're finished.

Cobalt and Xenon Only

Detail View Tool



This tool creates a detail view from a drawing view. Detail view geometry is associative to the geometry in original view and the detail dimension. Change the geometry within the area defined by the detail and the detail view automatically updates. Move the dimension location or the size of the detail in the original view and again the detail view reflects the changes. Detail view dimensions are placed on the Sheet View layer.

Tech Note:

The font type, size or style of the dimension label cannot be changed. Use the Text tool to create your own labels.

By default, details dimensions are alphabetical. If you place more than one detail view in your drawing, the detail view dimension and label automatically increment to the next letter. If you close the file containing detail views and later reopen it and add another detail view, the dimension and label increment to the next letter, based on the last detail label in the file. You can also enter any label text desired. However, user-defined label text does not automatically increment. The graphic here shows a drawing view and an associated detail view.

Once a detail view is created, you can move the detail view dimension in the original view from its default location, shown as A in the left graphic above. Select the label, choose

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Edit>Show Points and drag the control point/detail dimension to another location. The detail view label, shown as Detail A above, cannot be moved.

This tool can only be used when a drawing view is present.

Tech Note:

You cannot paste solids into section views. If you want to add a solid, you must add it to the original view.

Using the Detail View Tool

1. Select the Detail View tool. The Message Line reads: *Detail View: Pick drawing view for detail view.*

   If a view is already selected, skip to step 3.

2. Select the drawing view.

   The Message Line now reads: *Detail View: Pick detail center.*

3. Click the center point on your geometry for which you want a detail view.

   The Message Line reads: Detail View: Pick detail edge pt.

4. Click the detail edge point.

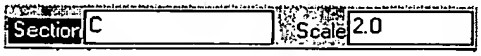
   The Message Line reads: *Drag window to final position.*

5. Drag the view to a new location and click the mouse button. Notice that as you move the pointer an outline of the frame appears.

   The detail view displays with the view label and the scale shown in the Status Line If you want a different label or scale, enter the data in the appropriate data field and press ENTER (Windows) or RETURN (Macintosh). The detail view dimension appears in the original view.

   If you want to delete the view later, you must delete both the view and the dimension line in the original view. You may have to adjust the detail labels to accommodate the deletion.

The Status Line contains the detail view label and Scale.



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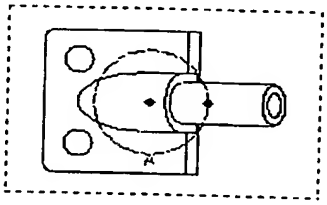
Resizing the Detail View Dimension

You can change the diameter of the detail view dimension. Select the detail dimension and choose ***Edit>Show Points***. Two controls point appear defining the center and edge of the dimension.



Select one of the control points and drag it to a new location.

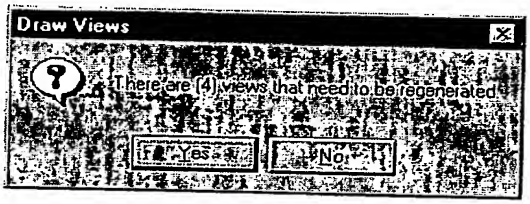
Manual View Regeneration Tool

This tool regenerates all drawing views on the sheet. This is useful if you have selected the Manually Regen option in a Draw View Properties dialog box for one or more views and made changes to them.

Using the Manual View Regeneration Tool

1. Select the Manual View Regeneration tool.

   The Draw Views dialog box appears telling you the number of views that need to be regenerated.



2. Click Yes to regenerate all views.

   Click No to close the dialog box without regenerating the views.

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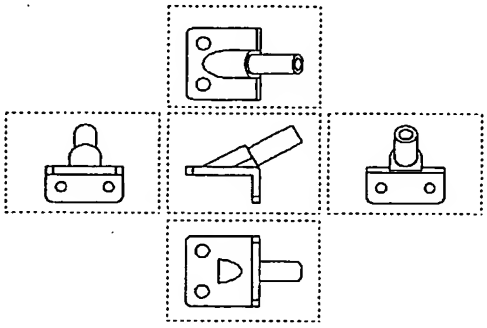
Unfolding Views

You can quickly create new views from an existing drawing view by using an unfolding operation. The graphic here shows a center view unfolded in four directions.



1. Choose the Selection tool and activate you want to unfold.
2. Hold down the CTRL key (Windows) or the OPTION key (Macintosh) and drag the view to the left, right, top or bottom of the view.

   A new view is created with the geometry rotated 90° from the selected view.

Editing a Drawing View

Adding Objects to a View

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You can add objects in an existing or new view by either creating them using a drawing tool or by copying and pasting them into a view.

Creating Objects in a View

Once a view is activated, you can create additional wireframe and solid objects in the view. Since the Drafting Assistant functions inside the view, you can place the objects relative to the other objects already present.

You can also place dimensions and text in the view using the Dimension tools and the Text tools. All dimensions are placed on the Sheet View layer rather than the Dimension layer for drawing views. Since the drawing view work plane is set to top, all text and dimensions appear correctly.

All entities created in an active view, display only in that view.

Copying and Pasting Objects into a View

You can also paste objects copied from another location into a selected view. When you paste the objects, this Designer Elements program examines the contents of the paste buffer and inserts it in the active view. The inserted object is associative to the original object. This feature is valuable when creating a new drawing view.

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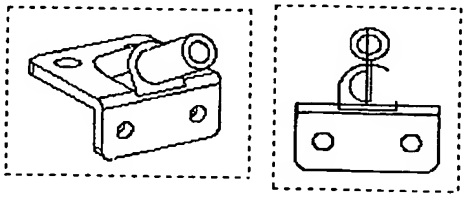

If the objects were copied from a 3D model, the 2D objects created from them are associative to the model. You can change the view later and the geometry will display correctly. If you copy 2D objects from a view (regardless of whether they were created from a 3D model), these objects do not reference the 3D model. Since this 2D geometry is not associative to a 3D model, changing the view may not create a complete view. The graphic here shows an example of this. The 2D geometry in the Trimetric view was pasted into the Right Side view.

Editing Objects in the View

Editing 2D Geometry created from 3D Geometry

You can edit your 2D geometry in a number of ways: changing the line characteristics, layers or modifying the geometry.

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Changing Line Patterns and Layers

Since an object placed in the drawing views are true 2D wireframe, you can assign a different line pattern, color, weight or layer to the entities making up the object. To change a line characteristic, select the line in the view. In the Pen menu, choose a new pattern, color or weight. You can also change line characteristics through the Edit Objects dialog box.

To change the layer for the line, choose Edit>Change Layer and select the new layer. You can also change the layer through the Edit Objects dialog box.

Modifying 2D Geometry

Because the 2D geometry created from the 3D model is composed of individual curves, you cannot modify the length or the location of the curve by dragging a control point like you would in a normal model. If this were possible, the associativity of the model would be destroyed. For example, a line in a model may represent the visible edge of a cylinder which is not actually present in the 3D model. If this was changed, you would no longer have a cylinder.

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If you attempt to modify the geometry, this Designer Elements program provides the following warning.

Given this, you have two ways to edit geometry, removing the links of the selected geometry or editing the 2D geometry by changing the parent 3D geometry using the Design Explorer and the Edit Objects dialog box.

- Select the object and choose ***Edit>Remove Links***. A warning appears reminding you that this command deletes all associative relationships and that you cannot undo this operation. Click OK and the object can now be changed.
- Select the object. Display the Design Explorer and open the history tree to show the parent geometry for the selected object. Double-click on that object to display the Edit Objects dialog box. Change the desired value and click OK. The 3D model and the 2D geometry updates. The graphic here shows a selected edge and the Design Explorer with the ACIS Solid parent.

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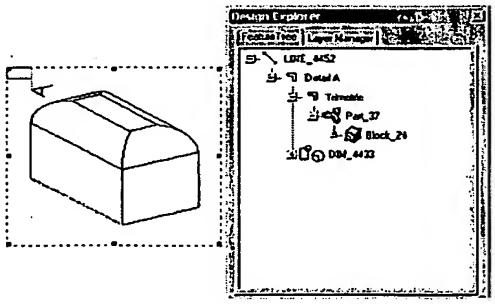

Editing View Characteristics

Changing the Scale of a View

You have two methods for changing the scale within a view:

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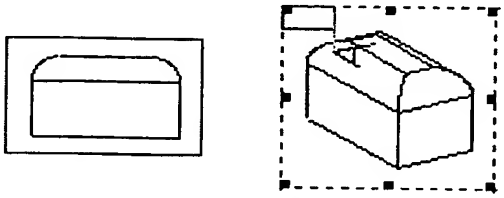
- Choose the Properties command in the Drawing View menu and change the scale value within the dialog box.
- Select the drawing view boundaries by dragging a selection fence around the view rather than clicking on the view. Choose ***Window>Edit Objects*** and in the Geometry page, enter a new scale. Clicking the view activates the view but does not select just the view boundaries. The left graphic here shows the selected view boundary. The right graphic shows an activated view.

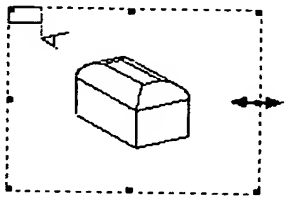


Select all drawing view boundaries in a layout by using the Selection Mask.

Resizing a Drawing View

You have three methods for resizing a drawing view:

- Dragging the view edge: Select the view and place the pointer on a control point of the edge you want to move. The pointer becomes a two directional arrow. Drag the edge to a new location. Place the pointer at a corner control point to resize two adjacent edges at the same time. The graphic shows the view control points and the directional arrow.



- Using the Properties command in the Drawing View menu
- Using the Frame to Extents command in the Drawing View menu

Dimensions and Drawing Views

If dimensions in the drawing view go outside the bounds of the view such that you can't see them, choose the Frame to Extent command in the Drawing View menu. You can also drag the view edge manually to completely display the dimension. See Chapter 27 for information on the standard dimension tools.

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All dimensions placed in drawing views, go on the Sheet View layer rather than the Dimension layer. This enables you to turn off all other layers and still print the sheet with views containing dimensions.

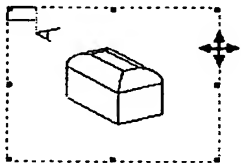
Changing or Deleting View

The view orientation, scale and view properties can be changed through the Drawing View menu for each view. If you change the view orientation of a drawing view and cannot see all of the geometry in the view, choose the Center command or Frame to Extents command in the Drawing View menu. See the "Drawing View menu" section earlier in this chapter for more information.

You can delete a view through the Drawing View menu. You can delete all drawing views at once by selecting the Drawing view type in the Selection Mask, double-clicking on the Selection tool and pressing the BACKSPACE key (Windows) or the DELETE key (Macintosh).

Moving a Drawing View

You have two methods for moving a drawing view:



- Dragging the View: Select the view. Place the pointer over a view edge, not a control point. The pointer becomes the move symbol. Select the top or bottom view edge to drag the view vertically. Select the left or right view edge to drag the view horizontally.
- Using the Properties command in the Drawing View menu

Drawing Views and the Edit Menu

If you notice that the Edit menu name is red as you work in a Sheet view, your drawing contains some unresolved links. This occurs when you move geometry or make some other change. Choose Edit>Resolve Links. The Edit menu name becomes black again.

Layout Templates

This Designer Elements program provides you with 14 layout templates. These include the following:

1ViewA (B, C, D or E).vs	Displays Top view on the specified size sheet.
4ViewA (B, C, D or E).vs	Displays four views, Top, Front, Right Side and Trimetric on the specified size sheet.

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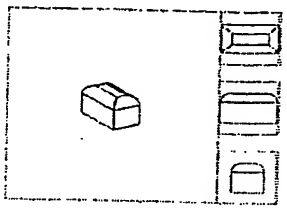
Design4.vs	Displays four views, Trimetric, Top, Front and Right Side in drawing views and arranged in a design view layout (as shown here).
Draft4.vs	Displays four views, Top, Front, Right Side and Trimetric in drawing views.
FrontRight.vs	Displays two views, Front and Right in drawing views.
FrontTop.vs	Displays two views, Front and Top in drawing views
Top.vs	Displays the Top view in a drawing view.
Trimetric.vs	Displays the Trimetric view in a drawing view.



You create change the views used in particular layout by using the Properties command in the Drawing View menu (see an earlier section) or create a layout with your own views. See the next section.

Creating your own Layout Templates

You can create your own templates from scratch or by modifying an existing template.

Modifying an Existing Template

All templates provided are Designer Elements program files that can be opened like any other file.

1. Open one of the files in the Layouts folder.
2. Make whatever changes you desire. You can adjust the format, add text to the title block, and add or remove drawing views.
3. Save the file under another name in the Layouts folder.

   You can now select this template from the Layout pull-down menu in the Model to Sheet dialog box.

Creating a New Template

You can create a new template from scratch using a standard Designer Elements program file.

1. Open a new file.
2. Set the view and plane to Top.

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3. Create the title block and border for the layout and add text if desired.
4. Using the Drawing View tool, add drawing views.
5. Save the file in the Layouts folder.

   You can now select this template from the Layout pull-down menu in the Model to Sheet dialog box.

Printing the Layout

If you want to print your layout with the views, turn off all layers other than the Sheet View layer. Make sure your Page Setup (Windows) or Print Setup (Macintosh) matches the layout.

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Page Setup and Printing

You've created all your geometry using the wireframe, surface and solids tools, conducted various operations on it such as extruding, sweeping and blending and added some basic annotation to your drawing.

All during your design process, you did not have to worry about scaling or your paper size. Now that you're ready to print a hard copy, you'll need to set up the page according to a scale, paper format, orientation and your printer specifications.

This Designer Elements program prints and plots on most printers and plotters supported by your computer. After you have followed the manufacturer's instructions for installing and setting up the printer or plotter, you can set up your page size as needed.

The following topics are covered:

- Drawing Size
- Printing a Drawing
- Print Window

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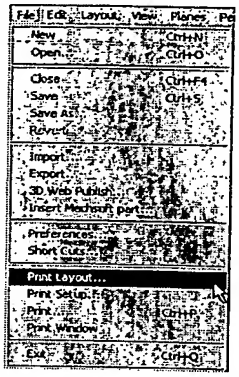
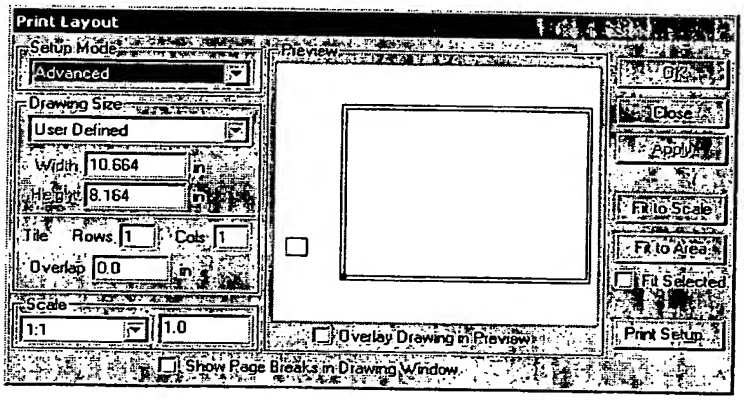
Print Layout Command

The *Print Layout* command, located in the File menu, displays the dialog box which allows you to set the page size, scale and other options.



When you select the command the Drawing Size dialog box appears. The graphic here shows the *Advanced* Setup mode.



The Drawing Size dialog box includes the standard buttons: OK, Close and Apply.

OK — Click this button to accept all changes and close the dialog box

Close — Click this button to ignore any changes made since you last clicked Apply and close the dialog box.

Apply — Click this button to accept and register all entries made in the dialog box without closing it.

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The Drawing Size dialog box includes the following sections: *Setup Mode, Drawing Size, Scale, Preview* and *Utility Controls.*

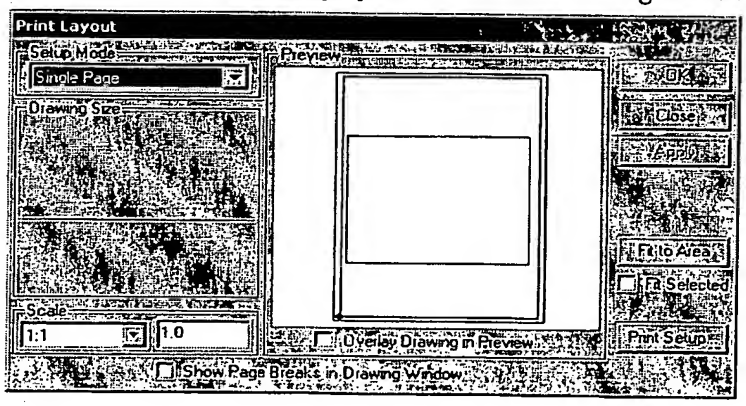
Setup Mode Section

The Drawing Size dialog box supports four setup modes; *Single Page, Height* and *Width, Rows and Columns* and *Advanced.*

For all setup modes, a drawing frame displays in the Preview window. The outer drawing frame represents the physical page size. The inner drawing frame represents the printable page area. The page settings are obtained from the current printer settings. Changed the printer settings by choosing ***File>Print Setup*** (Windows) or ***Page Setup*** (Macintosh).

Single Page Mode

The *Single Page* mode is the simplest printing mode. It is the best mode to use when printing to a large format plotter or when you need a quick single page plot. As the printer settings are changed, the drawing frame updates to conform to the new settings. Selecting this mode displays the most basic Drawing Size dialog box.



You can set the plot scale by making a selection from the Scale drop down list or by changing the value in the Scale data field.

Use the Fit to Area button to automatically compute the scale that will fit the drawing objects to the printable area.

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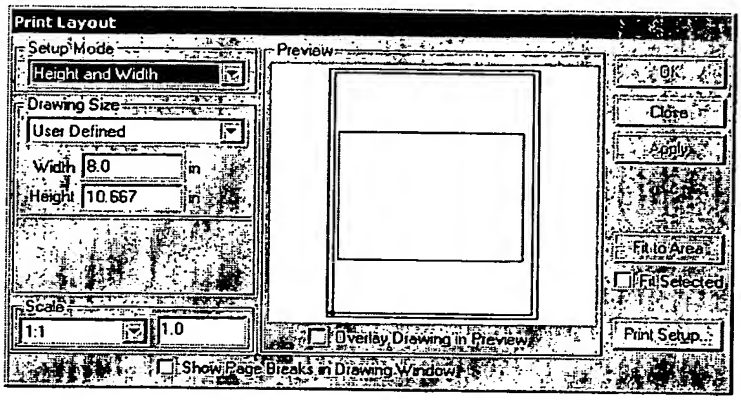
Height and Width Mode

The *Height and Width* mode is used to generate large standard or user defined plot sizes when using small format print devices (e.g. 8.5 x 11 laser or inkjet printer). Selecting this mode displays the format area of the Drawing Size section.



For drawing sizes larger than what your printing device allows, the drawing is tiled and can later be assembled into the large format plot. The *Preview* window displays the page tile edges within the drawing frame automatically.

You can specify a standard drawing size or a custom drawing size. You can set the scale in the *Scale* section or automatically compute the scale using the Fit to Area button.

Rows and Columns Mode

The *Rows and Columns* mode is used to force whole pages to be used for tiled plots. Unlike the *Height and Width* mode, this mode will use all the printable area

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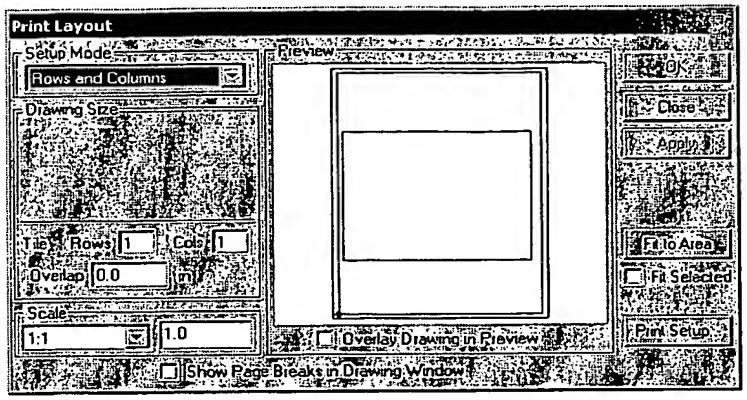
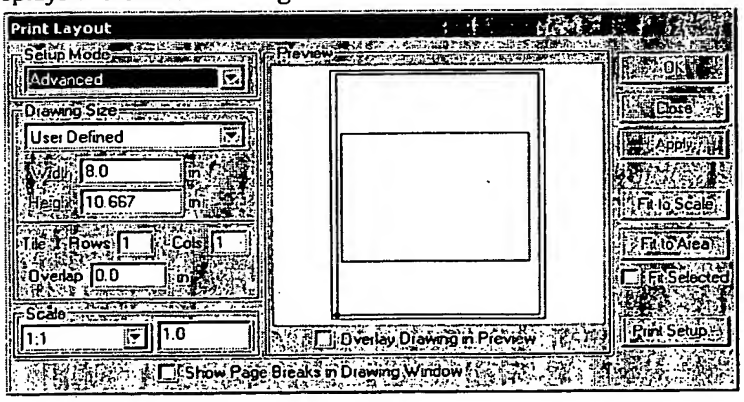
available for the plot. However, the plot will not be a standard size. Selecting this mode displays the *Tile* area of the *Drawing Size* section.



You can specify the tile rows and columns by changing the values in the *Rows* and *Cols* data fields. The Overlap data field controls the how tile pages will overlap. The overlap region helps align the pages when assembling the final plot.

Advanced Mode

The *Advanced* mode provides access to all height, width, rows and columns plot settings. This mode permits complete control over all aspects of tiling. Selecting this mode displays the entire Drawing Size section.



You can specify any of the listed elements.

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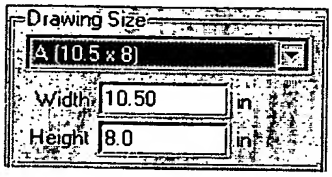
Drawing Size Section

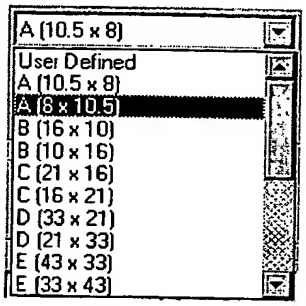
This section contains the drawing format sizes available and the height and width of the selected format size.



Tech Note:

When you choose a drawing format size, be sure that you set the page orientation for your printer to the same orientation selected in the Drawing Size dialog box. Choose ***File>Print Setup*** or ***Page Setup*** to display the Printer dialog box to check your current page orientation.

The format size field includes a pull-down menu listing all of the formats and their sizes.



You can choose any of the predefined sizes, A, B, C, D, E, F, G, H, J and K. Each format size includes a listing for portrait and landscape orientation.

The units (inches or mm) are determined by your preference setting. When you select one of these predefined formats, the size is displayed in the *Width* and *Height* fields.

You also have the ability to set your own drawing size by selecting the *User Defined* option in the list. When you choose this option, you enter the size in the *Width* and *Height* fields.

The drawing sizes displayed in the pull-down list are contained in the *DrawSize.ini* file in the Environ folder within the Designer Elements program folder. This file can be edited but keep it mind it may change or be overwritten by future Designer Elements program installations. If you do choose to edit the file, save the original version under another name before doing so. Then you will have a copy in case you want to return to the default sizes.

If the page size is larger than the size supported by your printer, values are automatically entered in the *Tile* area to accommodate the drawing and appears as such in the *Preview* window. (Choosing the *Advanced* mode shows the tiling specifics.) See the next section for more information on tiling.

Before choosing your size, you must determine what size format your printer or plotter can support.

Drawing size and page tiling are synchronized based on the scale. Values entered in the drawing size fields affect those in the page tiling and vice versa. *Height* affects

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Rows and *Width* affects *Cols*. The last field in which you enter values controls the drawing dimension, represented by the activated field name. The associated field name is unavailable.

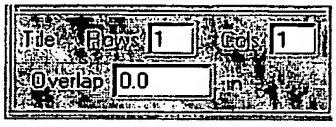
Setting the Format Size

1. Choose ***File>Print Layout.***
2. Choose the *Height and Width* mode from the pull-down menu.
3. Display the pull-down menu for the drawing format size.
4. Select the desired size.

   The size is displayed in the *Width* and *Height* fields.
   If you selected the *User Defined* format size, enter the size in the *Width* and *Height* fields.

Page Tiling

This *Tile* area of the Drawing Size section allows you to set up your file so you can print a larger drawing in tiled sheets.



This area includes these elements:

Rows

Represent the number used to print your file, based on the size supported by your printer driver and the drawing scale. A value automatically appears in this field when you select a drawing size larger than that which your printer supports.

If the exact drawing size is unimportant, just specify the number of rows. Since the number is synchronized with the *Height*, entering a different value changes the height of your drawing.

Cols

Represent the number of columns used to print your file, based on the size supported by your printer driver and the height of your drawing. A value automatically enters in this field when you select a drawing size.

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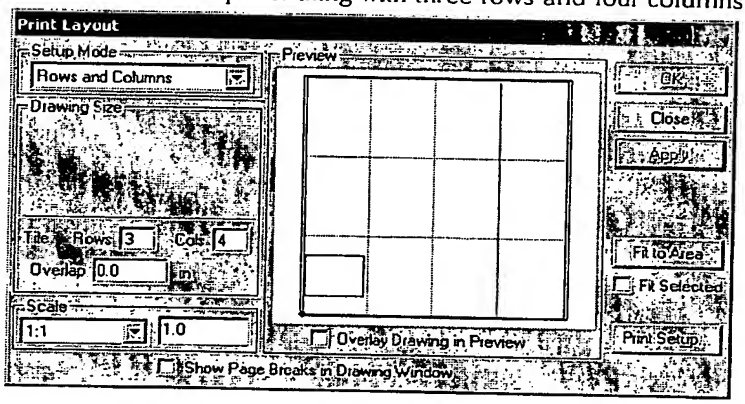
If the exact drawing size is unimportant, just specify a number of columns. Since the number of columns is synchronized with the *Width*, entering a different value changes the width of your drawing.

Overlap

When tiling, you can specify a page overlap (between 0 and .75 inch or 20 mm). The overlap determines how much of the geometry repeats on the right and top area of each tile page. The overlap region can be used to align the tiles when joining the pages.

This graphic shows an example of tiling with three rows and four columns.



Tiling operates independently of the page orientation.

Setting the Page Tiling

1. Choose ***File>Print Layout***.
2. Choose the *Rows and Columns* mode from the pull-down menu.
3. Enter the desired values in the *Rows* and *Cols* fields. The drawing size fields adjust accordingly.
4. Specify the overlap for the tiled pages. The units are determined by your preferences setting.

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Scale

This section allows you to specify the scale of your drawing. You can select a standard scale from the pull-down list or set your own scale in the data field.

Scale Options

The pull-down list provides you with these scaling options: User Defined, 5:1, 4:1, 3:1, 2:1, 1:1, 1:2, 1:3, 1:4 and 1:5.

Choosing one of the standard scales enters a value in the data field. A 5:1 scale, enters 5.0 in the field. A 1:5 scale enters a 0.20 in the field.

If you want to specify your own scale, enter the value in the data field. The scale name changes to User Defined, regardless of the scale entered.

The drawing scales displayed in the pull-down list are contained in the *Draw-Size.ini* file in the Environ folder within the Designer Elements program folder. This file can be edited but keep it mind it may change or be overwritten by future Designer Elements program installations. If you do choose to edit the file, save the original version under another name before doing so. Then you will have a copy in case you want to return to the default scales.

Setting the Scale

1. Choose ***File>Print Layout***.
2. Display the pull-down list for the scale.
3. Select the desired scale. The scale appear in the edit field.

   If you selected the User Defined scale option, enter the scale in the data field. The left graphic below shows the *Preview* window of a rectangle at a scale of 2:1. The right graphic shows the *Preview* window of an rectangle at a scale of

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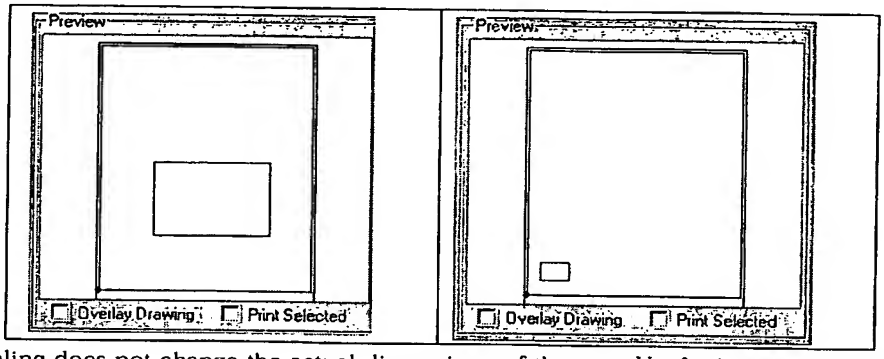
1:2.



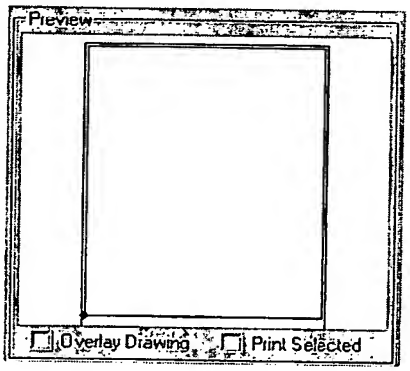
Scaling does not change the actual dimensions of the part. Verify this by selecting an object and choosing ***Window>Edit Objects***.

Preview Section

To assist you in choosing the correct format size for your drawing, This Designer Elements program includes the *Preview* section containing the Preview window and two check boxes, O*verlay Drawing* and *Print Selected*.

Preview Window

A preview of the drawing appears in this section of the Drawing Size dialog box.



The outer drawing frame represents the physical page size. The inner drawing frame represents the printable page area. The page settings are obtained from the current printer settings. If you change the drawing size, the *Preview* window still displays the last drawing size selected until you click Apply.

Only objects or part of objects that lie within the page bounds are printed.

The *Preview* window displays a rectangle, representing the drawing or the actual geometry. This display is determined by the *Overlay Drawing* setting. See the next section for more details.

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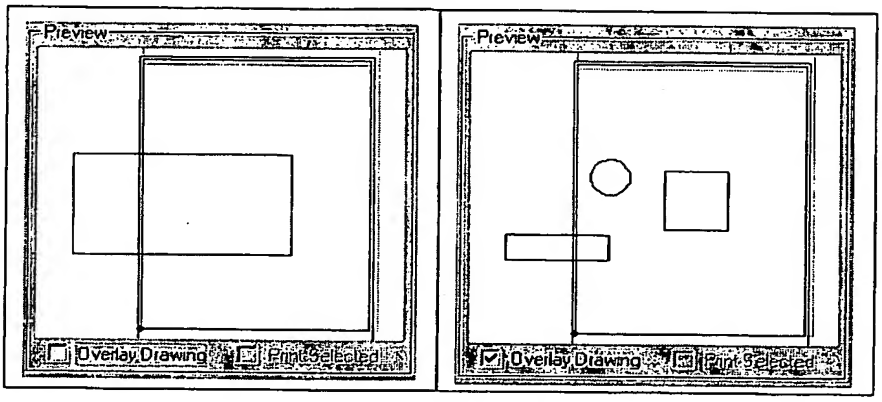
The Axis displays in the *Preview* window only if it's displayed in your drawing area. The Axis does not print.

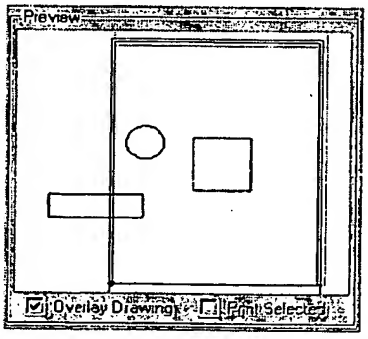
Overlay Drawing

This check box determines how your geometry appears in the *Preview* window. When the box is not checked, a red rectangle displays, representing the drawing area used by the objects (the graphic on the left below). When the box is checked, the actual geometry displays (the right graphic).



Print Selected

This check box allows you to specify which objects within the drawing bounds get printed. When the check box is empty, all geometry within the page bounds print. When the box is checked, only selected geometry within the page bounds prints and the view window zooms in on that area. In the graphic here, only the square prints.



In the graphic, all of the geometry displays. If you deactivate the *Overlay Drawing* check box, only the selected rectangle displays in the *Preview* window.

This check box only becomes available when you select geometry.

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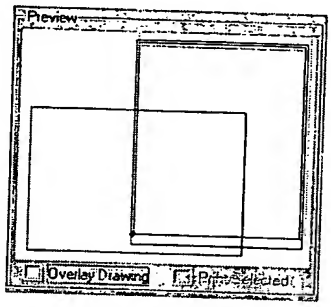
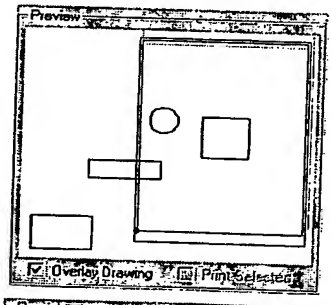
Preview Section Example

An example of using the option in this area might clarify how the *Preview* window and check boxes interrelate.

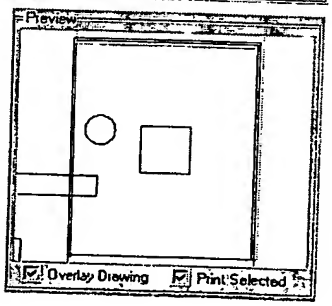
1. Draw some geometry.
2. Choose ***File>Print Layout***. The Drawing Size dialog box displays. The *Preview* window shows a red rectangle representing the object area.



3. Check the *Overlay Drawing* box to display the actual geometry.
4. Click OK in the dialog box to close it.
5. Select an object within the page bounds.
6. Choose ***File>Print Layout*** to display the dialog box again.



7. Check the *Print Selected* box. The *Preview* window zooms in on selected object.



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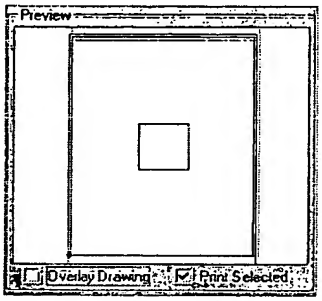
8. Click the *Overlay Drawing* check box to remove the check mark. Only the selected geometry displays.

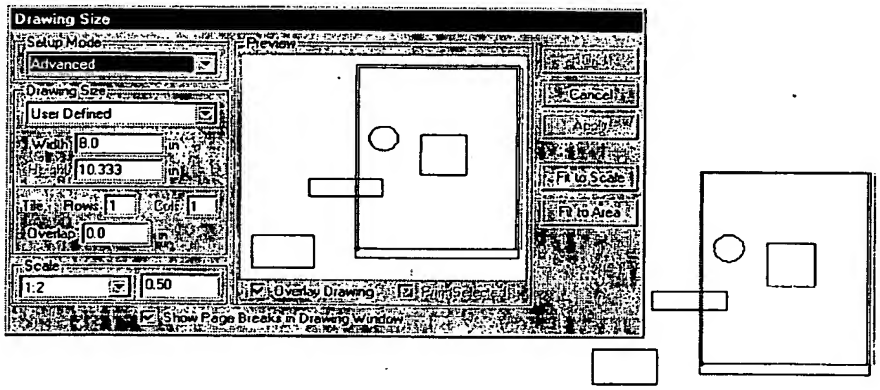


Utility Controls

The Drawing Size dialog box contains utility controls for setting up your drawing. These include the *Show Page Breaks in Drawing Window* check box, the Fit to Scale button and the Fit to Area button.

Show Page Breaks in Drawing Window

This check box allows you to specify whether to display the page breaks/boundaries in your drawing area. When this box is checked, the page bounds display.



When you click OK and close the dialog box, you can move the page boundaries. Place the cursor over the marker at the lower left corner of the page boundaries. The cursor becomes the move symbol (shown to the right). Drag the page boundaries to the new location. See the "Move the Print Boundaries" section at the end of this chapter for more information.

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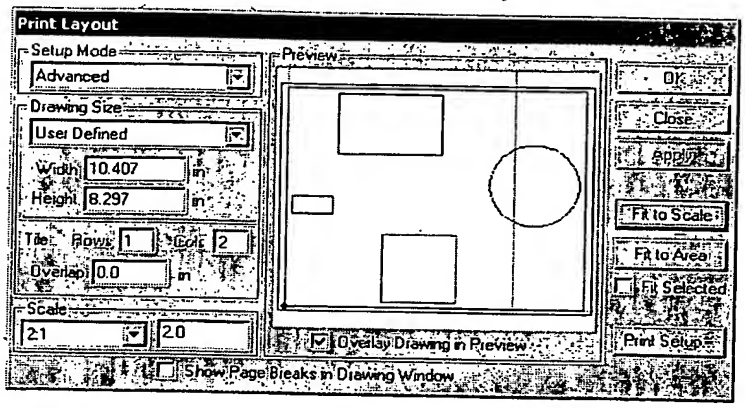
Fit to Scale

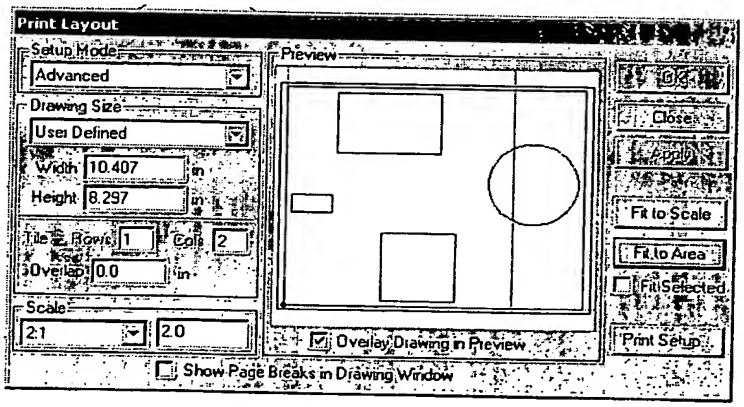
Clicking this button uses the current scale value and automatically changes the height, width and page boundaries to fit the geometry.



This button only displays with the *Advanced* mode.

Fit to Area

Clicking this button automatically computes the scale and page boundaries to fit the geometry to the entire tiled plot's printable area.



For a single page to print, set both *Rows* and *Cols* to 1 and press the Fit to Area button.

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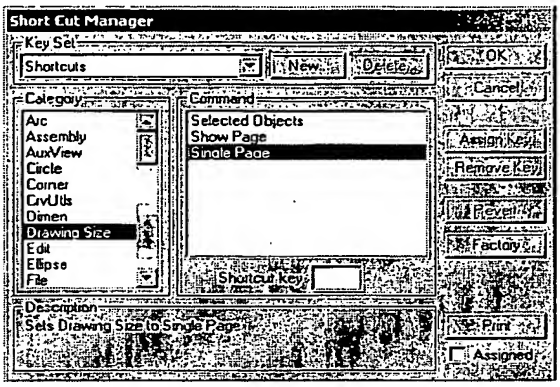
Drawing Size and Short Cut Key

A new short cut key, Single Page, was added to Drawing Size category in the Short Cut Manager.



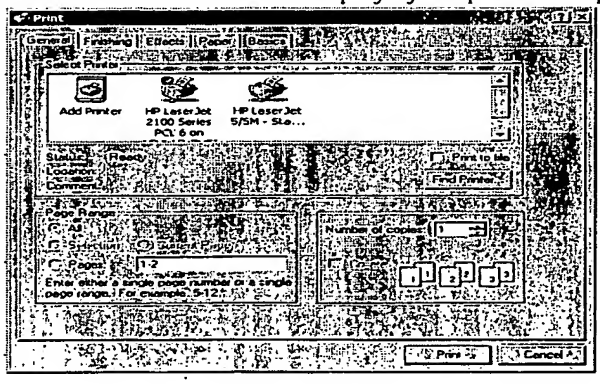
Using this key automatically chooses the *Single Page* mode, activates the *Fit to Area* function and *Shows Page Breaks*. You can assign whatever short cut key you desire.

Printing a Drawing

After you have set up your page you are ready to print. Choose ***File>Print Setup*** (Windows) or ***Page Setup*** (Macintosh).

Print Setup (Windows); Page Setup (Macintosh)

Choosing this command in the File menu displays your printer setup window.



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Choose the necessary settings for paper size and page orientation to agree with your settings in the Drawing Size dialog box. Click OK to save settings. See your printer manual for information about setting your printer options.

Print Command - CTRL+P (Windows); z +P (Macintosh)

This command in the File menu prints or plots the current document as specified in the Drawing Size dialog box.

The area printed or plotted is the portion that fits on the page size specified in the Drawing Size dialog box. Choose ***File>Print Layout*** to scale your drawing to the appropriate size and reposition the print/plot region.

You can specify tiling (printing on several pages to be pasted together) by choosing ***File>Print Layout>Advanced***.

Print to a File

You can print to a file rather than to a plotter or printer. In that way, you don't have to have a plotter attached to your computer. Someone else can plot the drawing without having this Designer Elements program on the plotter's computer. The type of plotter you choose when setting up the page determines the format of the plot file.

If you choose a PostScript printer, the file format is Encapsulated PostScript; use the HPGL language when you select Hewlett Packard plotters. The computer that finally plots the file must have an application compatible with the file format of your printer or plotter.

Plotter Font

When you are using a plotter, you should specify the Plotter font for the text and dimensions on your drawing. You can also generate special characters and accents as described in Appendix B.

Printing/Plotting Region

When you choose ***File>Print*** only the geometry within the page boundaries prints. You can view those boundaries by choosing ***File>Print Layout***. If your printer does not support the size, gray boundary lines are displayed in the window, representing the boundaries and the tiling feature activates. For all printers and plotters,

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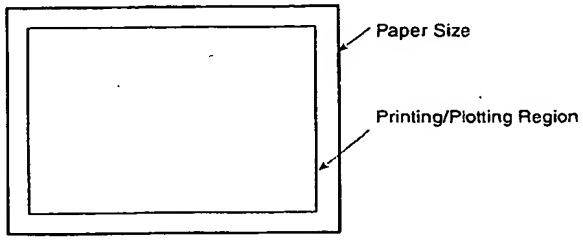
the plotting region is smaller than the actual page size because most printers and plotters cannot plot to the edge of the paper, allowing room for the margins.



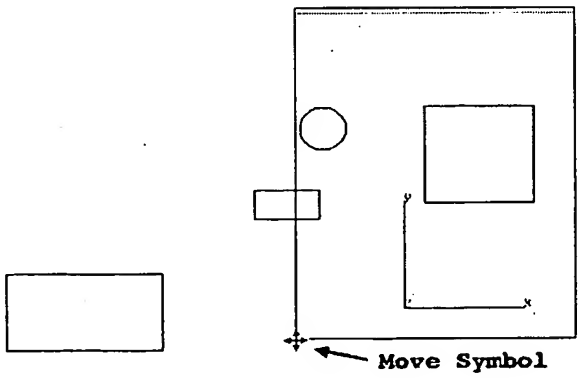
Vellum displays the Printing/Plotting region

The size of this region is based on the paper size and the printer or plotter driver currently selected.

Moving the Print Boundaries

If the geometry you want to print is not contained within the page boundaries you can move the page boundaries.

1. Choose ***File>Print Layout***.
2. Select the *Show Page Breaks in Drawing Window* check box.
3. Click OK. The dialog box closes and the page boundaries are displayed in your drawing.
4. Place the cursor over the marker at the lower left corner. It becomes the *Move* symbol.
5. Drag the boundaries to the new location.



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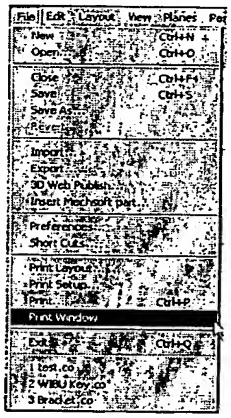
Printing and Rendering

This Designer Elements program prints both wireframe and rendered geometry. When you want to print rendered geometry, set the *Static Render* option in the Render Options dialog box to the desired mode.

If your printer supports color, *Static Render* must be set to Flat or Gouraud.

Print Window

The *Print Window* command in the File menu copies the image within the drawing screen and sends it to the printer.



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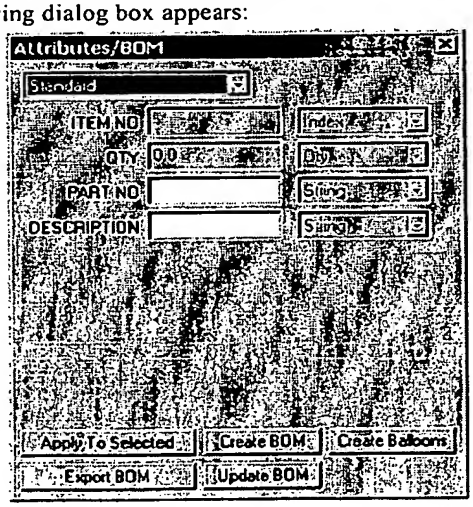
Attributes and Bill of Materials

User Attributes and Bill of Materials provide a means to apply custom attributes to entities and display that data in a table or export it to an external application such as Microsoft Excel.

This tool is located in the Window menu (***Window>Attributes and BOM***). After selecting this tool the following dialog box appears:



The Bill of Materials dialog box contains a drop down menu of predefined attribute templetes and five button options.

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The predefined templets are:

Material	Assigns a material to an entity.
Price	Assigns a price to an entity.
Standard	Assigns a part number and description to an entity.
Stock Size	Assigns part number, stock size and description to an entity.
Vendor No.	Assigns a part number, a vendor number and a description to an entity.
Perimeter	Calculates the 2D perimeter for curves.
Volume	Calculates the volume of a solid.
Weight	Calculates the weight of a solid using the material assigned in the ***Verify>Mass Properties*** command.
Area	Calculates the surface area of any polygon, surface or solid.
Area 2D	Calculates the 2D area and centroid properties for curves.
Mass Properties	Calculates the Mass Properties for a solid using the ***Verify>Mass Properties*** command.

The first five attribute templates require the user to supply all of the information associated with the attribute. The latter six automatically extract attribute information from the entity.

The five buttons on the Attributes/BOM dialog box have the following functions:

Apply To Selected	This options applies the current BOM attribute to the selected entities.
Create BOM	This option creates a Bill of Material Table using the attributes defined by the pull down menu. The user is

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prompted to enter text height, colun width and item order.

ITEM NO	PART NO	DESCRIPTION	CG-X	CG-Y	AREA
1		POLYGON_33	-5.187963	1.490709	2.082140
2		ELLIPSE_34	-5.379836	-0.656798	1.245559
3		POLYGON_35	-5.379836	-2.472215	1.530388

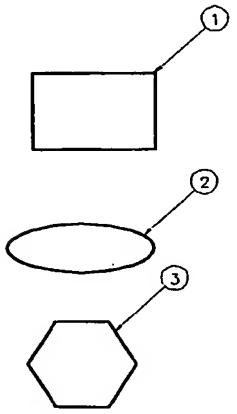
ITEM NO	PART NO	DESCRIPTION	CG-X	CG-Y	AREA
3		POLYGON_35	-5.379836	-2.472215	1.530388
2		ELLIPSE_34	-5.379836	-0.656798	1.245559
1		POLYGON_33	-5.187963	1.490709	2.082140

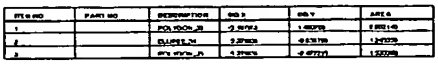
Graphic 2 equals Item Numbers Up

Tip:
The Bill of Materials function does not incorporate the use of Copy/Cut and Paste.

Create Balloons

This option adds balloon item dimensions to geometry. This option requires an item number type be used with the attribute. All of the supplied attribute files have item numbers.





Export BOM

This options creates a comma-delimited file that can be read into Microsoft Excell or other similar spreadsheet programs.

Update BOM

This option updates the BOM table and the corresponding balloons.

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BOM User Defined Templates

You can create user-defined templates by creating an attribute definition file. Attribute definition files are located in the BOM folder of the install directory. The first line of the file contains the attribute name. This is the name that will appear in the attribute pull-down menu. The next lines contains two columns, the first column contains the attribute definition string, and the second column the attribute type. Commas separates each column

The following attribute types are supported:

Float	User-assigned signed decimal value.
Integer	User-assigned signed integer value
String	User-assigned character string
Area	Calculates the entity area and assign.
Perimeter	Calculates the entity perimeter and assign.
Volume	Calculates the entity volume and assign.
Weight	Calculates the entity weight and assign.
Qty	Counts the number of occurrences this attribute is used.
Index	Index used to refer to the attribute in the BOM table or balloon callout.
Name	Extracts the name from the entity.
CG-X	Calculates the CG-X value from an entity.
CG-Y	Calculates the CG-Y value from an entity.
CG-Z	Calculates the CG-Z value from an entity.
Sum	Sums the value the previous attribute.
Material	Extracts the material type from the entity.

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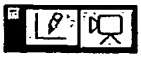
Parametric Constraints (Cobalt Only)



Cobalt support 2D profile dimensional constraints. This feature provides a mechanism for dimensioning entities to define distances and angles in order to establish geometric constraints between curves.

To access the Constraint tools and enter a 2D Sketch Mode go to the Window menu and choose Constraints.

Sketch Mode

To enter the 2D Sketch Mode in the Parametric Constraint tool you must first choose the sketch tool.

Clicking on this tool expands the tool palette, showing the tools available while in sketch mode.

By entering the sketch mode the application knows to adjust the user interface in such a manner that creating 2D sketches is intuitive. The sketch mode does the following automatically when creating a new sketch or modifying an existing sketch:

1. Hides tool palettes such as surface and solid modeling which have no meaning in the 2D sketch environment.
2. Adjusts the drafting assistant to always snap into the sketch plane and see only snaps in the sketch plane. This is accomplished by turning on the "Work Plane" and "Plane Only" options in the Snaps dialog box.
3. Turns on Auto Constraints. Auto constraints will automatically create geometric relationships (coincident, tangent, concentric, perpendicular) as you create and modify geometry. You can turn this off in sketch mode through the popup menu activated with right-click (Control + click on Mac).
4. Creates a set of layers for the sketch that includes the container layer, "Sketch 1" and sub layers that include construction, profile, constraints, and dimensions. Note the geometry in the construction layer is ignored if the sketch is used in profile-based operations such as skinning, sweeping, lathing, or extruding.
5. Show Dimming is turned on. This feature is useful when working on faces of solids where edges not in the sketch plane are dimmed. In addition, if you modify a sketch

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on the face of a solid, the part is rolled back to the point where the sketch was original created.

Exiting Sketch Mode

Exiting the sketch will restore the above settings back to the previous settings before the sketch was entered. One other advantage of the sketch mode is that it allows the ability to temporarily suspend the regeneration of a dependent feature. This means that while in sketch mode you can perform a series of operations such as adding and removing curves to the sketch that would otherwise invalidate downstream operations. However, when exiting the sketch mode, you must resolve a valid profile such that dependent features can then be updated.

The tools available in Sketch Mode are listed below.

Sketch Tool Creates or Modifies an existing sketch.

Auto Constraints Tool The auto constraint tool automatically applies constraints to curves and dimensions selected by the user.

This tool is particularly useful when working with data created outside the sketch tool. The auto constraints tool will add the following constraints to the selected geometry: Horizontal, Vertical, Tangent, Concentric and Coincident.

Horizontal Constraint Tool Adds a horizontal constraint to a line. You can select one or more curves to apply a horizontal constraint. Horizontal is defined by the work plane x-axis. The two chosen points will have the same x value when completed.

In this tool, note the message line:

When applying a horizontal constraint, pressing the CONTOL key (OPTION on Mac) will give the user the option to align the geometry horizontally to a specific point on another line. The two chosen points will, when completed, have the same x value.

Vertical Constraint Tool Adds a vertical constraint to a line.

You can select one or more curves to apply a vertical constraint. Vertical is defined by the work plane y-axis.

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In this tool, note the message line:

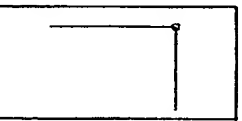
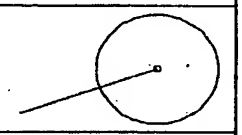
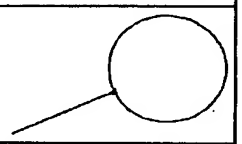
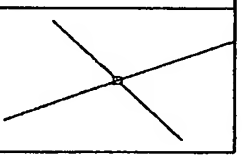
Vertical Constraint: Pick line for vertical constraint [Ctrl = Share Y position] [Shift = Extend]

When applying a vertical constraint, pressing the CONTROL key (OPTION on Mac) will give the user the option to align the geometry vertically to a specific point on another line. The two chosen points will have the same y value when completed.

Coincident Constraint Tool This tool adds a coincident constraint between two object positions.

Objects that are recognized for coincident constraints include lines, arcs, circles, ellipses, splines, points, and edges of solids. The point of coincidence to the object is automatically determined by use of the drafting assistant. Referencing end points, mid-points, vertex, centers, and point on are preserved. In the case of point on (point along curve) the coincidence constraint may be anywhere along the curve. All others are fixed to specific locations. The floating coincident constraint is represented by a small triangle symbol whereas a fixed constraint is a small rectangle about the two shared points.

Some examples of coincident constraints:

End Point/End point	
End point/Center Point	
Point On/End Point	
Midpoint/Midpoint	

The coincident constraint tool allows for several means of attaching relationships. The first method is to follow the user prompts:

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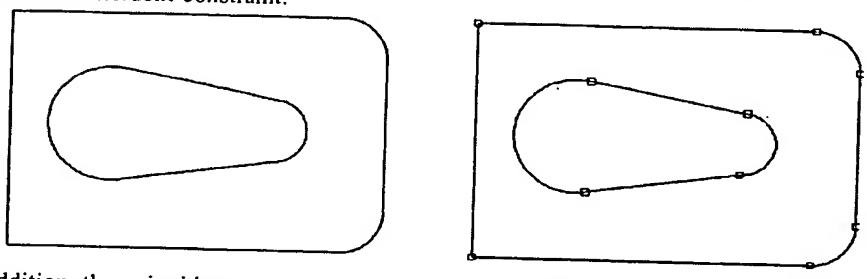
1. Pick the first curve for coincident constraint.
2. Pick the position along the first curve using the drafting assistant.
3. Pick the second curve for the coincident constraint.
4. Pick the position along the second curve using the drafting assistant.

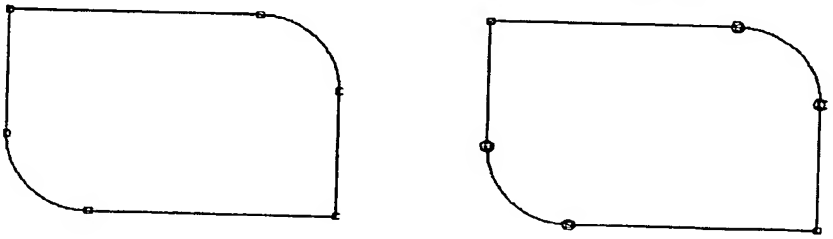
The second method for creating coincident constraints is to box select two or more curves at the first prompt. In this case all curve end points that lie within 0.001 inches will be applied a coincident constraint.



In addition, the coincident constraint tool allows for pre selections. If any curves are pre-selected before you hit the tool palette icon, coincident relations are automatically applied to the selected objects.

Tangent Constraint Tool This tool creates a tangent constraint between two or more curves.

To use the tangent constraint tool simply select the curves you would like to apply a tangent constraint. When selecting more than 2 curves, the tangent constraint tool only applies a constraint between curves that are already tangent within 1 degree. This constraint draws a circle to represent the existence of a tangent constraint.



Parallel Constraint Tool The parallel constraint tool adds a parallel constraint between two lines.

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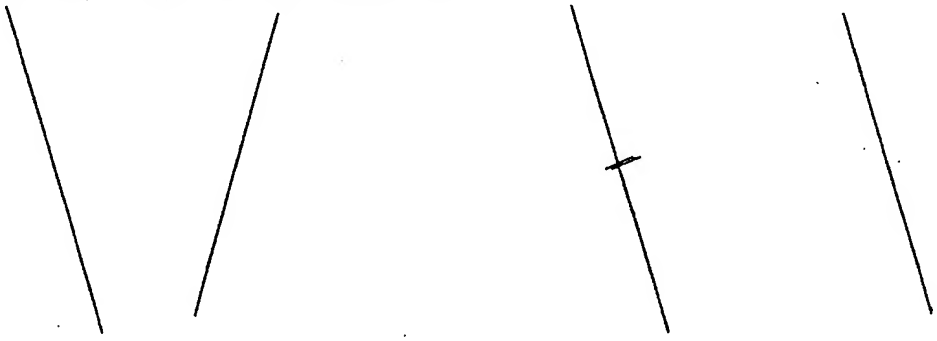
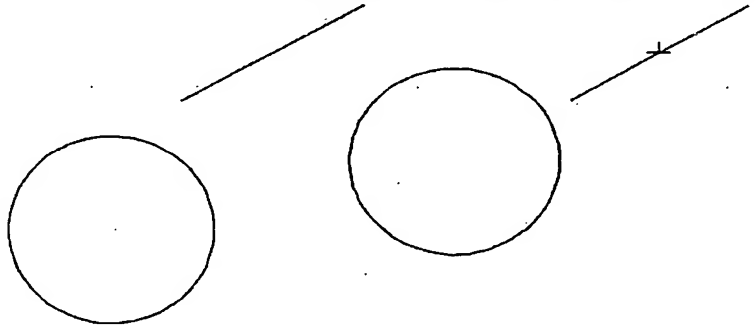
Only lines can be used in this tool, arcs, circles, ellipses, splines are ignored. The constraint symbology is two small parallel lines.



Note: The solver determines which line to move based on a set of rules involving the curve type, other curve relations and minimization of geometry movement. Therefore the order in which you pick the curves is irrelevant as to which curve is actually moved.

Perpendicular Constraint Tool The perpendicular constraint tool creates a 90° angle between a line and another curve.

In the example below a coincident constraint is needed to attach the line to the circle. As in other constraints the entity that moves is independent of selection order.



Fixed Constraint Tool The fixed constraint tool locks the entity from being moved by the solver.

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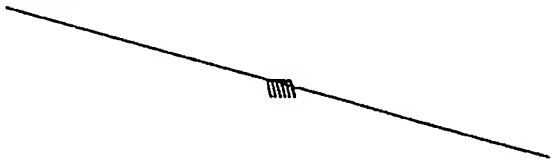
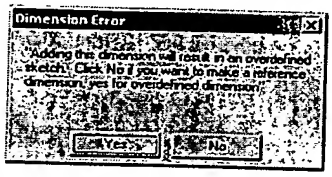
The user is allowed to move the entity with the move or translate tools. A fixed object uses the fixed display color which is by default gray and whose symbology consists of a collection of slanted lines.

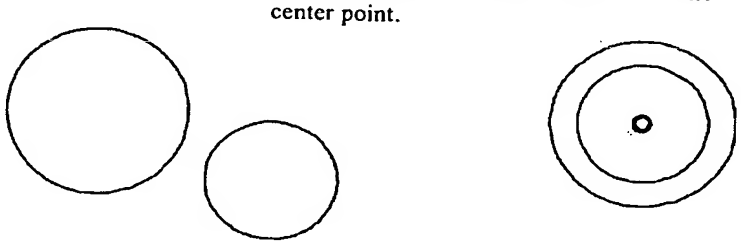


Note: Attempting to dimension a fixed entity will result in the display of the dimension error dialog box.



This error message implies that the position, direction, and length are fixed due to the fix constraint previously applied.

Concentric Constraint Tool The concentric constraint tool creates a constraint that forces circles to share the same center point.



This concentric tool recognizes circles and points as valid selectable entities.

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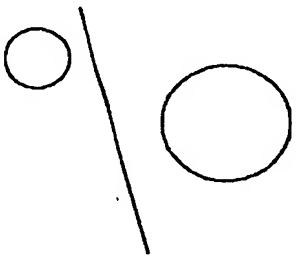
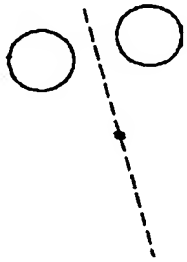
Symmetric Constraint Tool

This tool creates a symmetric constraint between entities of asimilar type and symmetry line.

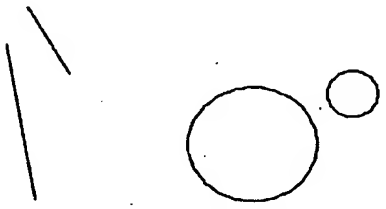


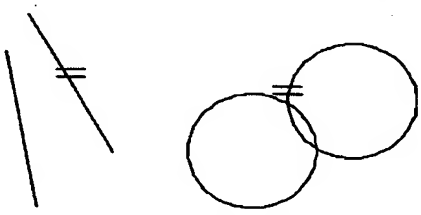


Note: Use the mirror tool in the transformation tool palette while in the sketch mode to automaticaly apply mirror constraints at the time the mirror operation is performed.

Equal Constraint Tool

The equal constraint tool applies an equal distant or radius constraint between two entities.





This tool works only with lines or circles. When selecting two lines, the lengths are forced to be the same for the two entities. In the case of circles, the same radius value is applied between the selected circles. As with many constraints, this operation is independent of which curve is selected first due to the method in which the solver finds solutions.

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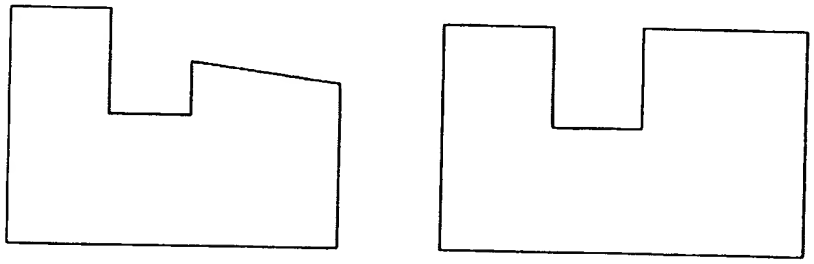
Colinear Constraint Tool

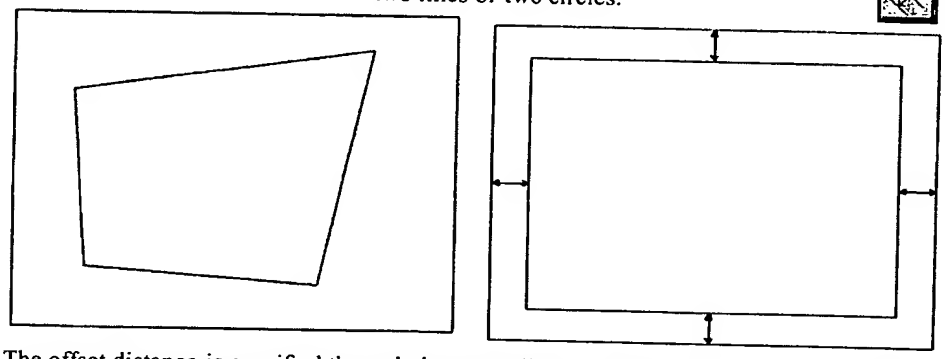
The colinear constraint makes two lines colinear.



Colinear implies that the two resulting lines have the same direction and line within the same line. It does not imply anything regarding their lengths.

Offset Constraint Tool

This tool adds an offset constraint between two lines or two circles.



The offset distance is specified through the status line. In addition you can use the edit objects dialog to change an existing offset value. Click on the constraint symbol to display this edit page.

Animating Dimensions Tool

The animation tool will animate a sketch by modifying a dimension value through a range of values. To use the animate tool, select a dimension that was used within a sketch and then choose the Animate Constraint tool.

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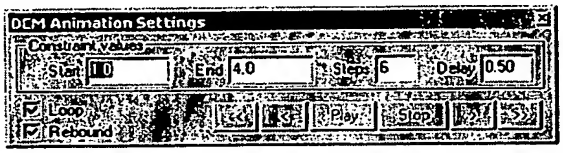
Note: If you animate a sketch while in sketch mode, only the sketch is updated. If you animate a sketch outside of the sketch mode, dependent surfaces and solids update accordingly.

DCM Animation Settings Dialog

Changing the settings

The DCM Animations Settings dialog box is used to control the behavior of a sketch dimension animation. The available animation settings are Start, End, Steps, Delay, Loop, and Rebound.



The Animation Settings dialog contains the following options:

Start	Specifies the starting dimension value.
End	Specifies the ending dimension value.
Steps	Controls the number of intermediate steps to use when transitioning from the dimension Start value to the End value. For example, if Start is 1.0, End is 4.0, and Steps is 6, the dimension will animate with the values of 1.0, 1.5, 2.0, 2.5, 3.0, 3.5, and 4.0. The delta value is computed by: (End - Start) / Steps.
Delay	Controls the number of seconds to wait between sketch dimension animation "frames".
Loop	If checked, the animation will continue to play indefinitely until stopped.
Rebound	If checked, the dimension value will transition from Start to End in Step steps, and then continue by "rebounding" from End to Start in Step steps.

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Running the Animation

Once the settings are made, select the dimension you want to animate. The animation will start automatically. You may control the execution of the animation using the VCR style buttons in the dialog box.

Deleting Constraints

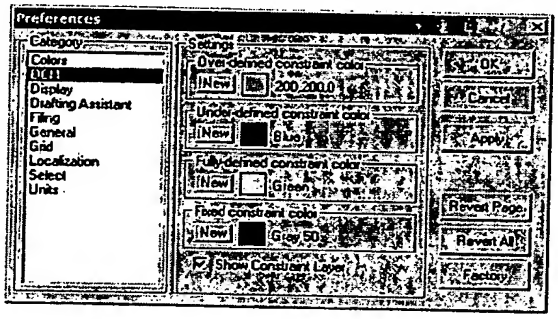
To delete any constraint, simply select the constraint symbol and hit the delete key or delete from the menu.

Verifying Constraint Relations

To confirm the entities used in a constraint relationship, simply move your cursor over the constraint symbol and pause for several seconds. After several seconds the entities involved in the constraint relationship will highlight in red.

Constraint Preferences

The Preferences dialog box under the File menu contains a category for DCM (Dimension and Constraint Management). The DCM Preference Settings includes options for setting:



Note: Avoid using the select color as a constraint color to minimize confusion between a constraint color and when an entity is selected.

Over-Defined Constraint Color Any entity in a sketch that has too many constraints or creates an ambiguity for the solver will display in the Over-Defined Constraint color. It is best to resolve any over-defined sketch entities before exiting a sketch.

Under-Defined Constraint Color Entities that still have degrees of freedom remaining are displayed in the under-defined constraint color.

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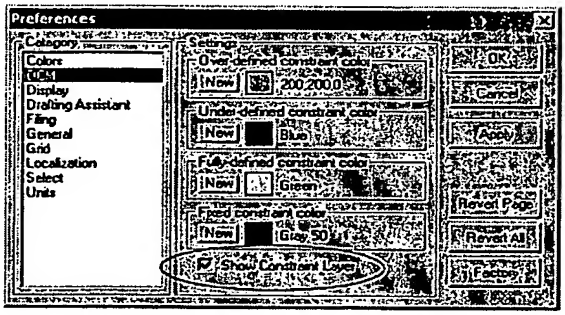
Fully Defined Constraint Color	Any entity whose position and size is fully defined is displayed in the fully defined constraint color.
Fixed Constraint Color	Entities that have a fixed constraint associated with their definition are displayed in this color.
Show Constraint Layer	When the show constraint layer check box is enabled, new sketches will automatically turn on the sketch layer. Sometimes complex sketches get visually complicated when all the constraint symbology is displayed.



Reference Edges

When in Sketch Mode, edges of geometry that are not on the sketch plane can be projected onto the plane. In sketch mode, simply select the Explode Edge tool:

Then click on the surface or solid edge you wish to project onto the sketch plane, and the exploded curve will appear on the sketch plane in the Fixed Constraint color. This color means that the object is associatively tied to another piece of geometry. Keep in mind that if you move the original geometry, the exploded curve will move as well.

Projecting a curve into a sketch

This is similar to exploding the edge of a surface or solid and projecting it onto the sketch plane, except that it will work for wireframe geometry. The procedure is the same as for Reference Edges. When in sketch mode, simply select the Explode Edge tool and choose the wireframe geometry you wish to project to the sketch plane. Once again, this geome-

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try is associatively tied to the original curve, and is displayed in the Fixed Constraint color.

Dimension-Driven Geometry

Dimensions created in the sketch mode are by default driving dimensions. This means that changing the dimensional value will force curves associated with the dimension to be updated to the new dimensional value. A dimension that is driven by the curve is called a reference dimension. Outside of the sketch mode, dimensions created are reference dimensions. To change a dimension from dimension-driving to dimension-driven, right click over the dimension.

Exiting Sketch Mode

Once a sketch is created and constrained, to exit Sketch Mode simply click the blue Exit icon in the lower right-hand corner of the drawing window:

Variables and Equations

This Designer Elements program supports the assigning of variables and equations to dimensions applied while in the Constraints Sketch Mode. When you have geometry that

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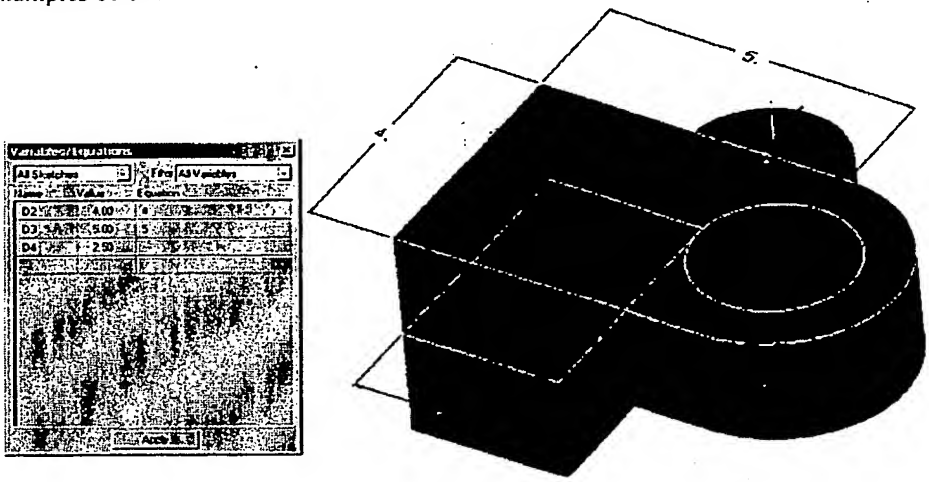
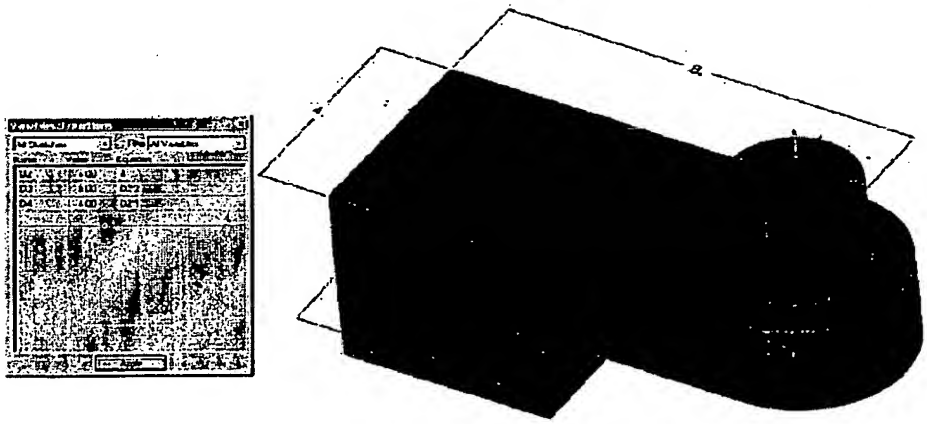
has been constrained using equations is an easy way to manipulate the geometry, see the examples below.





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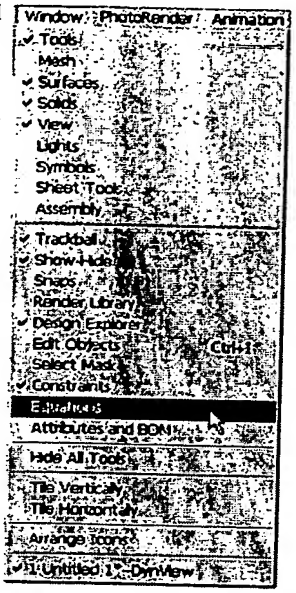
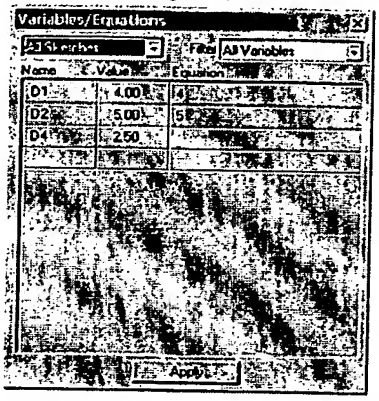
To open the Equations dialog box go to the Window menu and choose Equations (***Window>Equations***).



The equations dialog box supports all manner of mathamatical expressions. If you want to use one dimension as a reference to the other be sure to use its name from the name field as in the previous example. For a list of Mathematical operators that can be used see Appendix A.

Using Equations

Once you have completed your parametrically constrained part you may then edit the dimensions applied to the part using the equations dialog box, seen below.



The Equations dialog contains the following options:

All Sketches — This pull down menu contains the list of every sketch within the file.

Filter — Allows you to sort what type of dimensions are shown in the equations dialog box.

Name — The name of the dimension. You may change the name of the dimension by clicking once in this field and renaming the dimension.

Value — Shows the current value for each dimension. By pressing on the mouse while in the value field yo may high-

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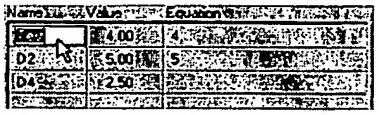
light that value on your screen. This is helpful when you have numerous dimensions in your sketches.

Equation — By clicking in this field you may enter the mathematical expression that will define the dimension.

Changing the Dimension Name

Use the following steps to change the name of the Dimensions.

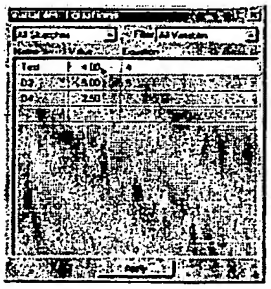


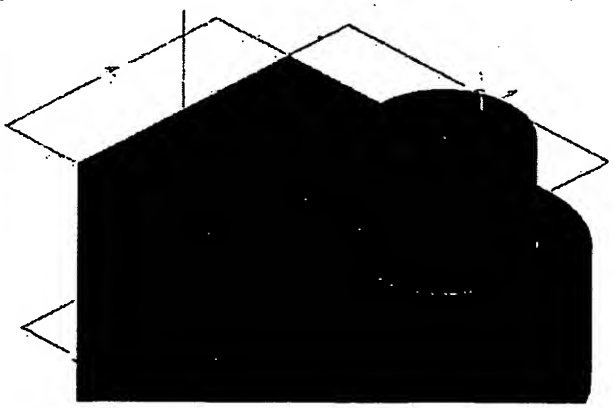
1. Move your cursor to the Dimension Name field you want to change.
2. Click once on the Dimension Name, the dimension highlights.
3. Change the name.

Highlighting Dimensions in the Sketch

Use the following steps to highlight the dimension on the screen.

1. Move your cursor to the Dimension Value field you want to highlight
2. Click and hold down the mouse button
3. The dimension highlights.





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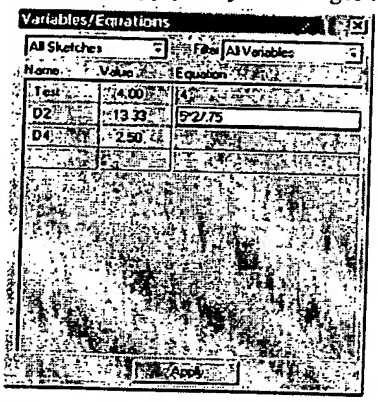
Adding Equations

Using mathematical expression or operators you can change or resolve the size of your part.

1. Click in the equation field of the dimension you would like to change.
2. Add your mathematical expression, such as D1*2/3.
3. Click Apply for your changes to take place.



Note: You cannot use the undo (Ctrl or Cmd + Z) for expressions applied in the equations dialog box. Repeat the steps above to change the dimension.

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Appendix A: Mathematical Operators

This Designer Elements program data fields accept the following mathematical operators.

Addition	x+y
Subtraction	x-y
Multiplication	x*y
Division	x/y
	x%y - modulo division which returns the remainder of x/y. The resulting value will always be from 0 to y-1.
Angles	this Designer Elements program accepts angle entries like the following: 45d30m30s, 45d30"30', 45d30', 30"30', 30m30s, 30"30s and 45°30"30
Absolute Value	abs(x) - absolute value of x
Arc tangent	atan (degrees)
Arc sine	asin(degrees)
Arc cosine	acos(degrees)
Cosine	cos(degrees)

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Sine	sin(degrees)
Tangent	tan(degrees)
Trigonometry	trig functions
Smallest Larger Integer	ceiling(x), or ceil(x) - returns a value representing the smallest integer that is greater than or equal to x.
Largest Smaller Integer	floor(x) - returns a value representing the largest integer that is less than or equal to x.
Degrees to radians	dtor(degrees)
Radians to degrees	rtod(radians)
Factorials	factorial(x), or fact(x) - factorial of x. (e.g. fact(4) = 4*3*2*1)
Logorithms	log(x)
	ln(x) - base 10 log of x, natural ln of x
Exponentials	exp(x) - e to the power of x where ln(exp(x)) = x.
	x^y - exponentiation (x to the power of y)
Negatives	neg(x) - negative of x
π	pi
Round	round(x) - rounds to the nearest whole value. For example, round(1.49) = 1.0, and round(1.51) = 2.0.
Random numbers	rnd - random value between 0.0 and 1.0
Square root	sqrt(x) - square root of x
Remove Fractional Part	truncate(x)
	trunc(x) - truncates to the whole value. For example, trunc(1.01)= 1.0, and trunc(1.99) = 1.0.

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Appendix B: Special Characters

You can use special characters and accents which are available with your computer. Usually these characters are described in an appendix of the user manual. Many symbols and characters are from the keyboard.

Windows:

Unlock the keypad with the NUM LOCK key and then hold down the ALT key and enter the numeric code for the character you want.

The character appears in the current font (including the DE Plotter font). The following list includes common symbols which are available in all fonts:

Accent	Key Combination
°	*ALT 0176*
ø	*ALT 0216*
±	*ALT 0177*

Tech Note:

Numeric values must be entered with an enhanced 101 keyboard using the separate number pad.

Macintosh:

To see the characters associated with a particular font, choose *Key Caps* from the Apple menu and select the font from the Font menu. When you press the z (Command), OPTION, SHIFT, or CONTROL key or a combination of these keys, the keyboard graphic changes to show which characters are available.

In addition to those characters you see in *Key Caps*, you can use the OPTION key to generate international accents in the current font, including the DE Plotter font.

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Accent	Key Combination
é	*OPTION+e* then letter to be accented
è	*OPTION+`* then letter to be accented
ñ	*OPTION+n* then letter to be accented
ü	*OPTION+u* then letter to be accented

The following list includes common symbols which are available in all fonts.

Accent	Key Combination
°	*OPTION+ SHIFT 8*
ø	*OPTION+ O* (letter or number)
±	*OPTION+SHIFT =*

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Appendix C: DXF/DWG Translator

This Designer Elements program supports the AutoCAD DXF (drawing exchange file) and DWG format. DXF files are standard ASCII text files. DXF files can easily be translated to other CAD systems which likewise support the DXF format. DWG is the native AutoCAD file format. This Designer Elements program supports the following DXF/DWG entities:

Read DXF/DWG	Description
3DFACE	Four points defining the corners of a face.
POLYLINE	A string of vertices defining a line.
LINE	Two points defining a line.
CIRCLE	Center point and circle radius.
ARC	Center, radius, start angle and end angle.
POINT	One point location in space.
TEXT	Collection of characters defining text string.
BLOCK	Symbol.
ACIS DATA	Curves, surfaces, solids generated by ACIS.
ELLIPSE	Valid for R13+.
LIGHTWEIGHT POLYLINE	Valid for R14+.
MULTI-LINE	Valid for R13+.

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SOLID	A filled region defined by 4 points.
SPLINE	Smooth spline curve valid for R13+.
LAYERS	Layer names, colors and locking.
LINE TYPES	Line font definition.

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Appendix D: IGES Translator

This Designer Elements program supports importing and exporting IGES files. The list of supported entities appears below.

Importing

This list provides the IGES Entity number, Form and IGES Entity name.

IGES	*Form*	*IGES Entity Name*
#100	0	Circular Arc
#102	0	Composite Curve
#104	0	Conic Arc General
#104	1	Conic Arc Ellipse
#104	2	Conic Arc: Hyperbola
#104	3	Conic Arc: Parabola
#106	11	Copious Data 2D Path
#106	12	Copious Data 3D Path
#106	63	Copious Data: Closed 2D Curve
#108	1	Plane Entity Bounded Face
#110	0	Line Straight
#112	0	Parametric Spline Curve

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#114	0	Parametric Spline Surface
#116	0	Point
#118	1	Ruled Surface
#120	0	Surface of Revolution
#122	0	Tabulated Cylinder Surface
#123	0	Direction Vector
#124	0	Transformation
#126	0	Rational B-Spline Curve
#128	0	Rational B-Spline Surface
IGES	***Form***	***IGES Entity Name***
#130	0	Offset Curve
#140	0	Offset Surface
#141	0	Boundary Entity
#142	0	Curve on Parametric Surface
#143	0	Bounded Surface
#144	0	Trimmed Surface
#186	0	MSBOBody
#190	0	Plane Surface
#112	0	Rt. Circular Cone
#194	0	Rt. Circular Conical Surf
#196	0	Spherical Surface
#198	0	Toroidal Surface
#502	1	Vertex List
#504	1	Edge List
#508	1	Loop
#510	1	Face
#514	11	Shell

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Exporting

The list provides the number, this Designer Elements program/ACIS name, IGES Entity number, Form and IGES Entity name for exporting or writing out an IGES file.

VS/ACIS	IGES	FORM	IGES ENTITY NAME
Straight	#110	0	Line
Ellipse	#100	0	Circular Arc
	#104	1	Conic Arc-Ellipse
	#126	0	Rational B-spline Curve
	#128	0	Rational B-spline Surface
	#120	0	Surface of Revolution
Plane	#190	0	Plane Surface
	#122	0	Tabulated Cylinder
	#109	0	Plane - Unbounded
Cone	#192	0	Cylindrical Surface
	#194	0	Conical Surface
	#120	0	Surface of Revolution
Sphere	#196	0	Spherical Surface
VS/ACIS	**IGES**	**FORM**	**IGES ENTITY NAME**
	#120	0	Surface of Revolution
Straight	#110	0	Line
Torus	#198	0	Toroidal Surface
	#120	0	Surface of Revolution
Point	#116	0	Point
Vector	#123	0	Direction
Transf	#124	0	Transformation
Vertex	#502	1	Vertex
Edge	#504	1	Edge

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Loop	#508	1	Loop
	#102	0	Composite Curve
	#106		Copious Data
		11	2D Path
		12	3D Path
		63	Closed Planar Curve
	#142	0	Curve on Parametric Surface
Face	#510	1	Face
	#144	0	Trimmed Surface
Shell	#514	1	Shell
Lump/Body	#186	0	MSBO

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Appendix E: Short Cuts

These are the short cut keys currently programmed in this Designer Elements program. There are many short cuts actions in this Designer Elements program which you can program with specific keys. Choose ***File>Short Cuts*** to display the Short Cut Manager. See Chapter 6, "Preference Settings," for information on how to program the keys.

Short Cut - Key	Action
Layout:Dec Display...{	Makes the previous layer visible and turns off all other layers except the work layer.
Layout:Inc Display...}	Makes the next layer visible and turns off all other layers except the work layer.
Layout:Isolate Layer...i	Brings up the Isolate Layer dialog box for setting the work layer.
Line:HorzConst...H	Creates a moveable horizontal construction line at your pointer tip. Move the pointer to the desired location and click to place the construction line.
Line:VertConst...V	Creates a moveable vertical construction line at your pointer tip. Move the pointer to the desired location and click to place the construction line.
Snap Alignments...A	Toggles the Drafting Assistant alignment snaps, like align x, y and z, between on and off.

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Snap Centers...C	Snaps to the center of an object.
Snap:Edges...E	Snaps to the edges of an object.
Snap:Faces...F	Snaps to the faces of a solid.
Snap:Intersections...I	Snaps to the intersections of objects.
SnapOnOff...Q	Toggles all Drafting Assistant snaps between on and off.
View:Front View...s	Changes the view to Front.
View:ISO View...f	Changes the view to Isometric.
View:Redraw Screen...r	Redraws the screen.
View:Side View...a	Changes the view to Side.
View:Top View...d	Changes the view to Top.
View:TRI View...g	Changes the view to Trimetric.
View:Zoom All...e	Activates the *Zoom All* command.
View:Zoom In...]	Activates the *Zoom In* command.
View:Zoom Out...[	Activates the *Zoom Out* command.
View:Zoom Window...w	Activates the Zoom Window command allowing you to drag a selection fence to specify the zoom area.

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Appendix F: Spline Text Files

You can create text files for importing Spline data into this Designer Elements program.

Creating a Text File for Importing a Spline

1. Use a text editor, a word processor or a spreadsheet to create a text file.
2. Input X, Y and Z values for your spline coordinates.

   The text file should be tab or space separated. Each line ends with a return. Line feeds after each return should have no effect.

   The text file should conform to the following columnar format:

1	1	0
2	2	0

   You are allowed to specify decimal coordinates as well:

1.33	1.1	0
2.4	2.5	3.5678

   Be sure that you press ENTER (Windows) or RETURN (Macintosh) after the last coordinate. If not, this Designer Elements program will not import the coordinates specified in the last line.
3. Save the file as Text only and import into this Designer Elements program using the Spline import format to create the spline.

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Appendix G:
Shader Attribute Definitions

This appendix defines shader attributes for all shader types. Although these terms may be used elsewhere in this Designer Elements program, these definitions only apply to the shader attributes.

Some shader attributes share a common base word like base color and decal color. In these instances, only the base word is listed and defined here. In this example mentioned, color is defined.

Information for these definitions was taken from LightWorks 5.0 Online Reference.

Color Class Attributes

amplitude	Sets the magnitude of an attribute relative to another.
axis	Sets the location of the axis for the attribute.
axis direction	Sets the direction for the axis when applying the simple wood shader.
bands	Sets the total number of bands around the evaluation cylinder when analyzing a surface.
brick height	Sets the brick height.
center	Sets the center of an attribute.
color	Sets the color of the attribute.

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color array	Sets the colors used for curvature divisions.
coverage	Sets the ratio of the area covered by the bands to the area not covered for a surface evaluation.
curvature	Sets the degree of curvature to be mapped.
curvature division	Sets the number of color divisions used for a curvature evaluation.
curvature type	Sets the type used in a geometric curvature evaluation. You have three types: gaussian, mean and absolute.
decal texture space	Sets the texture space for the attributes. Checking the Edit button displays a copy of the Render Material Settings dialog box from which you can set the space.
decal transparency	Sets the transparency from clear to opaque.
detail	Sets the complexity of the texture where a value of 1.0 results in a simple pattern and higher values result in a finer pattern.
draft angle	Sets the draft angle required to pull the object out of a mold.
file name	Sets the file name containing the image used for the shader.
fuzz	Sets the band sharpness.
gnarl	Sets the random roughness of the regular rings inside the trunk.
grain	Sets the intensity of the random grain effect where 0 (zero) equals no grain.
max cut off	Set the maximum curvature value for evaluating an object.
min angle	Sets the angle of the normal along the cylinder axis that defines the cylinder length and thus the reflection on the surface you are evaluating. Smaller values create longer cylinders.

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min cut off	Sets the minimum curvature value for evaluating an object.
mix	Sets the mix ratio of attributes.
mortar size	Sets the mortar size.
noise	Sets the visibility of the tree rings for the simple wood shader.
offset	Sets relative displacement of odd and even rows of the wood pattern.
plank length	Sets the plank length of the wood shader.
plank variation	Sets the brightness variation between wood planks.
plank width	Sets the plank width for a wood shader.
point on axis	Sets the point on the axis of the tree from which the wood is taken.
pull direction	Sets the direction the object will be pulled from the mold.
radius	Sets the radius of the attribute.
replication type	Sets the pattern of the wrapped image.
ring fuzz grain	Sets the intensity of the high frequency random roughness for the ring edges.
ring fuzz in	Sets the sharpness of the inner ring edges near the trunk center.
ring fuzz out	Sets the sharpness of the outer ring edge.
scale	Sets the scale of the attribute.
separation	Sets the distance between centers of adjacent spheres for the solid polka shader.
size	Sets the attribute size.
softness	Sets the softness of the feature used to define the shading.
strips	Sets the count of the rectangle wood planks.

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tolerance angle	Sets the degree tolerance added to the draft angle that allows the object to be pulled from a mold but with difficulty.
trunk direction	Determines the direction of the trunk axis.
trunk center	Specifies the center of the trunk.
vector	Sets the direction of the attribute calculated in determining the appearance of a particular shader.
vein contrast	Sets the color contrast of the marble veins where larger values produce a greater contrast.
width	Sets the attribute width.

Displacement Class Attributes

amplitude	Sets the magnitude of one attribute relative to another.
blend	Sets the size of the blend between the sphere and the surface for the shader.
center depth	Sets the depth of the spheres used for the dimple shader.
dented threshold	Sets the relative contributions made by the displacements and indentations for the casting shader.
detail	Sets the complexity of the texture where a value of 1 results in a simple pattern and higher values result in a finer pattern.
file name	Sets the file name containing the image used for the shader.
frequency	Sets the wavy or curving quality of edges.
irregularity	Sets the pattern shape from a square to an irregular convex shape.
radius	Sets the radius of the attribute.
scale	Sets the scale of the attribute.
separation	Sets the distance between the centers of adjacent spheres for the wrapped dimple shader.

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sharpness	Sets the sharpness of surface irregularities.
smooth max	Sets the maximum smoothness of the edges when using the leather shader.
smooth min	Sets the minimum smoothness of the edges when using the leather shader.
softness	Sets the softness of the feature used to define the shading.

Reflectance Class Attributes

absorption	Sets the amount of light absorbed.
ambient factor	Sets the amount of ambient light reflected.
amplitude	Sets the magnitude of one attribute relative to another.
bias	Sets the contribution of the two thread directions for the wrapped woven anisotropic shader. A bias of 0.0 causes all reflectance to be provided by threads along one axis. A bias of 1.0 causes all reflectance to be provided by threads along the other axis.
chrome factor	Sets the amount of chrome light reflected.
color	Sets the color of the attribute.
cylinder distance	Sets the distance between cylinders for the wrapped anisotropic shader. The distance determines the degree of anisotropy of the surface. A distance of 0.0 results in an isotropic (normal) reflection. A distance of 2.0 results in the maximum anisotropy.
decal texture space	Sets the texture space for the attributes. Clicking the Edit button displays a copy of the Render Material Settings dialog box from which you can set the space.
diffuse factor	Sets the amount of diffuse light reflected.

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exponent	Sets the sharpness of the specular reflection highlights.
file name	Sets the file name containing the image used for the shader.
floor height	Sets the height of the floor across the cylinders used for the wrapped anisotropic (grooved) shader. A value of 0.0 gives equals no floor. A value of 1.0 creates a flat, isotropic surface.
height	Sets the height of the attribute.
metallic layer factor	Sets the contribution of the metallic layer to the reflectance of the multilayer paint shader.
metallic flakes	Sets the metal for the metal flakes in the paint shader.
mirror factor	Sets the contribution made by light reflected in the mirror direction.
reflectance	Sets the reflectance of the shader.
refraction	Sets the amount of light refracted.
roughness	Sets the sharpness of the reflectance. Smaller values, such as 0.1, produce a sharper reflection.
scale	Sets the scale of the attribute.
selector	Sets the shader used to calculate the decal reflectance with respect to the base object. Clicking the Edit button displays a copy of the Render Material Settings dialog box from which you can set the shader.
shader	Sets the shader.
sharpness	Sets the sharpness of the surface.
softness	Sets the softness of the feature used to define the shading.
specular factor	Sets the amount of specular light reflected.
translucency factor	Sets the degree of translucency.

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transmission factor	Sets the amount of light that passes though the shader.
transparency	Sets the transparency of the shader.
width	Sets the width of the attribute.

Transparency Class Attributes

coverage	Sets the degree that an attribute covers the object with the associated shader.
color	Sets the color of the attribute.
detail	Sets the complexity of the attribute.
edge falloff	Sets the transparency edge falloff rate.
file name	Sets the file name containing the image used for the shader.
fuzz	Sets the degree of fuzziness for the edges.
grid size	Sets the size of the grid for the wrapped grid shader.
height	Sets the height of the attribute.
noise density	Sets the density of the roughness or irregularities.
scale	Sets the scale of the attribute.
s fuzz	Sets the softness of the s edge of the square for the wrapped square shader. The letter "s" is an identifier used to refer to one side of the square.
size	Sets the size of the attribute.
s max	Sets the maximum s dimension of the square for the wrapped square shader. The letter "s" is an identifier used to refer to one side of the square.
s min	Sets the minimum s dimension of the square for the wrapped square shader. The letter "s" is an identifier used to refer to one side of the square.
softness	Sets the softness of the feature used to define the shading.

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t fuzz	Sets the softness of the t edge of the square for the wrapped square shader. The letter "t" is an identifier used to refer to one side of the square.
t max	Sets the maximum t dimension of the square for the wrapped square shader. The letter "t" is an identifier used to refer to one side of the square.
t min	Sets the minimum t dimension of the square for the wrapped square shader. The letter "t" is an identifier used to refer to one side of the square.
transparency	Sets the transparency of the shader.
width	Sets the width of the attribute.
zero angle	Sets the angle between surface normal and view direction.

Texture Space Class Attributes

aspect ratio	Sets the ratio of the texture space which is defined as one unit of its height divided by one unit of its width.
axis direction	Sets the direction of the axis.
center point	Sets the center point of the cylinder used for mapping a texture space.
origin	Sets the origin point of the texture.
scale	Sets the scale of the attribute.
scale along axis	Sets the factor that an image is scaled along the axis.
scale around axis	Sets the factor that an image is scaled around the axis.
vector	Sets the direction of the attribute.

Background Class Attributes

angle	Sets the angle (radians) over which the environment map is sampled for each background pixel

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	allowing blurring. A 0 (zero) angle (default) means that the pixel's center point determines the color.
color	Sets the color of the attribute.
detail	Sets the complexity of the attribute.
distance	Sets the distance that the infinite planes are in front and back of the eye point. The two background images are placed on these infinite planes.
extrapolation	Sets how the background will cover the background area. You have three options: none, smear and tile.
file name	Sets the file name containing the image used for the shader.
intensity	Sets the brilliance of the reflection to be altered. The color is calculated for each background pixel by multiplying it with the intensity.
keep aspect	Sets the use of the pixel aspect ratio for the image. Images are automatically scaled to fit the viewport. Selecting True preserves the pixel aspect ratio.
keep texture	Sets the use of the texture for the image. Selecting True results in the image and file name attributes being referenced when no texture has been created yet.
missing ratio	Sets the ratio for mixing two shaders.
rotation	Sets the angle the image is rotated. The value must be either +/-90°, +/-180°, or +/-270°. Positive angles rotate the image clockwise. Negative angles rotate the image counter-clockwise.
scale	Sets the scale of attribute.
shader	Sets the shader.
softness	Sets the softness used to define the shading.

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Foreground Class Attributes

AA level	Sets the depth used for determining the number of samples. A value of zero results in no oversampling. A value of one results in two times the number of samples as the maximum.
AA threshold	Sets the limit value used to determine whether additional sampling calculations are performed to determine the light effect. If the scattered light at two sample points differ more than the threshold value, additional sampling occurs.
amplitude	Sets the magnitude of one attribute relative to another.
bounds	Sets the use of light scattering boundaries for calculating the effect when unable to determine where the light source contribution is negligible. Selecting False results in no bounds.
bounds on	Sets the use of light scattering boundaries for calculating the effect when unable to determine where the light source contribution is negligible. Selecting False results in no bounds.
bounds volume	Sets the volume of the boundary sphere used to calculate the scattering effect when unable to determine where the light source contribution is negligible.
color	Sets the color of the attribute.
density	Sets the density of the attribute.
distance	Sets the distance used in calculating the closeness of the fog to the viewer.
eccentricity	Sets the ellipse eccentricity used when calculating the light scattering effect for the Henyey-Greenstein scattering model. It has no effect on any other light scattering model. A zero eccentricity results in an isotropic scattering. A positive eccen-

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	tricity results in a forward scattering. A negative eccentricity results in a backward scattering.
error bound	Sets the limit for using the max lod attribute when calculating the detail effect using the scattering medium shader. The calculation time can become excessive depending on the detail level. This value determines the trade off point between calculation time and accuracy. The suggested range is between 0.0 and 1.0.
falloff threshold	Sets the spherical area of influence of the light sources. Beyond a certain area the light contribution would be negligible, making a large number of samples unnecessary. The default threshold is 0.001. The value's effect depends on the size of the scene and light source intensity. Thresholds that are too high result in spotlight clipping.
far	Sets the maximum distance for the foreground shader. Distances greater than the far value display the full background color.
flake size	Sets the size of the flake for the snow shader.
fog height	Sets the sets fog decrease rate.
ground normal	Sets the normal for the ground fog shader.
ground point	Sets the ground point for the ground fog shader.
ignore background	Sets whether the fog effect is applied to the background. Selecting True results in no background fog effect.
max depth	Sets the maximum distance used in calculating the light scattering effect. A smaller depth, near 0.0, results in an image lacking any volumetric effects. The default depth is 1000. The value's effect depends on the size of the scene.
max lod	Sets the maximum detail level for calculating scattered light using the scattering medium shader.
medium ambient	Sets a uniform light scattering through the medium.

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medium attenuation	Sets the attenuation within the medium or how it absorbs light along the way.
medium density	Sets the density of the medium through which scattering occurs.
medium shadows	Sets whether the medium received shadows.
min lod	Sets the minimal detail level for calculating scattered light using the scattering medium shader.
near	Sets the minimum distance that a background color appears. Distances less than the near value will not display the color.
noise gain	Sets the contrast in the noise. High values result in sharp transitions while low values result in smooth transitions.
noise octaves	Sets the number of octaves (frequencies/scales) used to determine the detail of the noise in the light scattering.
samples	Sets the number of samples taken to calculate the atmospheric scattering of light. A higher number results in greater accuracy but requires more calculation time.
scale	Sets the scale of attribute.
scattering model	Sets the model used for scattering the light. You have five options: isotropic, Rayleigh, Mie hazy, Mie murky and Henyey-Greenstein. Choosing Mie murky results in strong anisotropic forward scattering as would appear when looking directly at light sources.
source attenuation	Sets the falloff value for the attribute. Small changes in the attenuation value greatly affect the light scattering effect. Values are typically between 0.1 and 0.5.

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Glossary

Accelerators — Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.

Align — These commands let you align objects, including: text along the left sides, right sides, tops, bottoms, centers horizontal, centers vertical, to grid and equally spaced vertically.

ACIS — This Designer Elements program is based on this kernel, developed by Spatial Technologies.

Ambient Light — This light source provides equal illumination on all sides independent of the light source normal.

Annotation — Text on drawings, including notes, crosshatching and dimensions.

Alignment Angle — The angle of the Drafting Assistant's automatic construction lines. The specification is set in the ***Window>Snaps*** submenu.

Ambiguity Popup — This popup menu appears when you attempt to select one object among objects so you can choose the desired object.

Anchor — This point defines the direction when placing a distant light source in your drawing.

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Anti-Alias Feature Following	This feature performs a ray trace operation to bring out small geometric details and produce smoother images.
Arrange	This command in the Layout menu allows you to change the display of overlapping objects in your drawing.
Arrow Tool	Used for selecting objects to be operated on with subsequent commands. Also used to move selected geometry.
ASCII	An acronym for American Standard Code for Information Interchange.
Aspect Ratio	Refers to a mesh surface and specifies the maximum ratio between triangle edges.
Associativity	A link between an object and its dimensions or parent/child objects. In the case of dimensions, if the object is changed, the dimensions automatically change to match. In the case of parent/child objects, if the parent is changed the child also changes.
Attenuation	The reduction of light intensity with the distance from the source.
Attributes	The data fields associated with a particular object that define that object. This includes such as Layer Name, Pen Style, Color, X-Y-Z Coordinates. This is also a page in the Edit Objects dialog box.
Auto Heal Bodies	This function finds collections of surfaces that define closed volumes and convert them into solids. This occurs when importing Vellum 3D and IGES files into this Designer Elements program.
Auxiliary View	A view created from its parent view at the geometry location that the user specifies.
Axis	Displays the current view orientation of the X, Y and Z axis in the center of your screen.

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Bezier Curve	A free form curve. NURB splines are a superset of Bezier curves.
Blend	This is the filleting and rounding of solid edges.
Boolean Tools	These tools allow you to add, subtract and intersect solids.
Border	A frame showing the boundary of a view.
Boss	This is a cylinder extending from a solid and filleted at the intersection of the two.
Boundary	The geometry that defines the limits for operations such as trimming and relimiting.
CAD	An acronym for Computer-Aided Design.
CADD	An acronym for Computer-Aided Design and Drafting.
CADD.LIN	All line patterns are stored in this file in the Environ folder.
CAE	An acronym for Computer-Aided Engineering.
CAM	An acronym for Computer-Aided Manufacturing.
Case	This refers to the text case options in this Designer Elements program and include lower case, UPPER CASE and Title Caps.
Center Mark	A center-line dimension for circles and arcs.
Chamfer	A beveled or sloping edges between two objects.
Characteristics	See Attributes.
Child	An object created from another object or an operation performed on an object.
Circumference	The distance around a circle along its edge: =2 Pi r
Circumscribed	Enclosing a circle. In circumscribed polygons, the *midpoint* of each side of the polygon touches an imaginary circle (i.e. the polygon exactly surrounds the circle).

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Clamped Linear — Light intensity that diminishes according to the following formula, *Intensity/(distance+1)*.

Clamped Quadratic — Light intensity that diminishes according to the following formula, *Intensity/(distance^2+1)*.

Click — To press and release the mouse button. When you are told to click an object, move the pointer to the object and press and release the button.

Clipboard — The memory buffer where selections are stored when the *Cut* or *Copy* command is used.

Conic — These objects are used in aerospace design field and create curves defined by start point, end point, shoulder and slope control points.

Construction Lines — Lines, displayed as dotted or gray lines, that you use for exact alignment. The Drafting Assistant creates dynamic, temporary construction lines. You can also create permanent construction lines, which can be used in the geometry or used for alignment and then deleted.

Control Point — The *endpoint* or *midpoint* of an object or "knot" point defining a spline. The Drafting Assistant indicates these positions when the pointer is moved near them.

Coon Patch — A nurb surface with three or four sides.

Coordinates — Positions on axes that specify the point locations. Two-dimensional objects have x,y coordinates; three-dimensional objects have x,y,z coordinates.

Coplanar — This refers to objects that lie in the same two dimensional plane.

Copy — The command that places a duplicate of the selected geometry on the Clipboard. See the descriptions for the Polar Duplicate and Linear Duplicate tools and Transformation tools for additional copying methods.

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Cover Surface	A surface created from a profile.
Counter Bore	A hole created from a hole and a bore where the bore is a straight sided cylinder with a diameter larger than the hole. The hole extends from the end of the bore into the solid to complete the counter bore.
Counter Sink	A hole created from a hole and a sink. The sink is an angled hole with a diameter larger than the straight sided hole. The hole extends from the end of the sink into the solid to complete the counter sink.
Crayon Picker	This is a Macintosh color display allows you to chose a color from the crayon box.
Custom Colors	(Windows only) This button in the color display allows you to define 16 additional colors to show in the partial color display.
Cursor	The I-beam position indicator in the text tool and boxes which use text. Elsewhere, the position indicator is called a pointer.
Curvature	This command in the Verify menu displays a porcupine plot of selected curves or surfaces representing the direction and order of magnitude of the curvature.
Curvature Plot	This analysis displays geometry in a Gaussian Curvature plot which is the product (K_1 - K_2) of the principle curvatures at a point on a surface.
Curve	A line, circle, arc, ellipse, or spline.
Cut	The command to delete selected entities. The selection is placed on the Clipboard and can be pasted into the same or different documents or into documents created by other applications.
Cutout	A profile that has been extruded through a solid and removes all intersecting material.

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CYMK Picker	This is a standard color wheel for the Macintosh with the option to specify CYMK values.
Dashed	A line pattern made up of dashes.
Default	Built-in settings that are used by the system if you do not specify your own value or choice.
Defining Points	The X, Y and Z coordinates for the specified points of splines, mesh and slab primitives.
Deformable Faces	These are solid faces whose shapes can be modified by applying a gain pressure.
Degenerative	An object such as a surface or chamfer where the its length at the beginning and/or ending is equal to zero.
Delete	The command to erase selected geometry. The selection is not placed on the Clipboard; however, it can be retrieved within the limits of the *Undo* command.
Delta	A change, usually in position.
Design Explorer	This command displays a dialog box with the history tree of the selected object. It includes its own submenu of editing commands.
Detail View	An enlarged view of a specific area of the geometry displayed in a drawing view.
Dialog Box	A specification box that appears in response to certain commands. A dialog box allows you to provide information that qualifies the execution of those commands.
Diameter	The distance across an arc or circle, passing through the center.
Differential Scaling	Refers to the ability to use different scaling values for along different axes of a curve.
Dimension	A graphic object that displays the distance between two points. A measurement of an object.

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Direction	This command in the Verify menu displays the direction of the normals of an object.
Distant Light	This light source emits parallel rays of light, as if from a very distance source.
Divide	To segment a line or curve into equal parts. This is also one of the Trim tools.
Double-click	To press and release the mouse button twice, quickly.
Drafting Assistant	A unique feature which displays feedback notations and construction lines to aid with snap, alignment and constraint operations. The Drafting Assistant facilitates exact construction without requiring you to be exact. When the pointer is close enough to display feedback, the Drafting Assistant locks onto the exact location for you.
Drag	To press and hold the mouse button, move the pointer to a new location, and release the button.
Draft	A taper or a specific angle applied to a solid.
Draft Angle	The subtraction or addition of material at a specific angle along a solid. Draft angles are used to facilitate the placement or removal of a part from a mold. Positive draft angles add material. Negative draft angles remove material.
DXF	An acronym for Data Exchange Format-a format of AutoCAD files.
Drawing	A drafted, 2-dimensional drawing of a model. Usually a drawing including an orthogonal view.
Drawing View	A view containing 2D geometry created from a 3D model after choosing the *Model to Sheet* command.
Dynamic Render	The feature that displays your geometry in the selected render mode as you rotate the geometry.
DWG	AutoCAD's native file format.

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Endpoint	The first and last point of a line or curve. The Drafting Assistant indicates these positions when the pointer is moved near the *endpoint* of an object.
EPS	Encapsulated PostScript format for printing to a PostScript printer and for importing into compatible applications.
Estimate Count	Refers to a button in the Mesh Parameter dialog box which calculates the approximate number of facets based on specified parameters. The dialog box appears when you using the *Change Object Type* command and convert a surface or solid to mesh.
Export	To save a document in a file format that can be used by a different application program.
Extrude	Creates a 3D object out of a 2D profile.
Expression Parsing	Mathematical, trigonometric and exponential operators that can be used in the Status Line.
Face	A surface of a solid.
Facet	A way of representing surfaces in DXF and DWG files prior to release 13.
Falloff Angle	The angle that controls the sharpness of a spot light's edge.
Falloff Rate	This light distribution for a spot light from the center of the spot light cone to the outer edge.
Feature	A set of operations that may add material to or subtract material from your solid including blending, chamfering, creating holes, bosses, cutout and protrusions.
Field of View	The view angle for a perspective.
File	An individual document.
Fill Color	The color applied to a selected pattern for a smart polygon.

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Fill Pattern	The pattern applied to a smart polygon.
Fillet	An arc of a specified radius tangent to entities.
First Blind	An option for the hole tools which will extend a hole to the first open face.
Flatten View	This command in the Drawing View menu flattens 2D geometry within a view onto the Sheet View layer. This operation breaks the 2D geometry's associativity to the 3D model.
Flat Shading	Renders your object with a painter's algorithm using constant shading techniques.
Flavor	The types of IGES files this Designer Elements program can import and export.
Flip Normal	When this option is checked in the Render Options dialog box, the normals of an object are flipped. If light normals are pointed away from the view when rendered, the object will appear dark.
Font	The assortment of type used in text.
Gain	Pressure applied to a solid face to deform it.
Geometry	The objects used to construct parts.
Geometric Characteristics	Characteristics that make up the geometry of an object like, length, radius, Defining Points and Rho.
Gouraud Shading	Renders your geometry based on calculated light intensities at each vertex. It shades more quickly but with a lower quality than Phong shading. This rendering method uses Open GL (Windows) or QuickDraw 3D (Macintosh).
Gouraud w/Edges Shading	Renders your geometry based on calculated light intensities at each vertex and displays the face edge boundaries in a specific color.
Gregory Surface	A Nurb surface with more than four sides.

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Grid	The rectangular array of lines that facilitates measurement and alignment. The grid display can be turned on or off, and the spacing can be specified through the Layout menu.
Group	To specify several entities as one unit that will be treated as a single object.
Helix	A spiral curve.
Hidden	A line pattern used to draw lines that would not be visible in a solid part. Drafters traditionally use hidden lines for geometry that is behind other geometry.
Hidden Shading	Renders your geometry such that only visible edges are displayed.
Hidden w/Dimmed	Renders your geometry such that the visible edges are displayed and the hidden edges are dimmed.
Hot Spot	The point on the wireframe and dimension tool cursors that indicates the next point to click. The hot spot is represented by a dot with crosshairs on the cursor.
Histogram	A bar graph representing the frequency of a curvature smoothness (change in a curve over the change in curvature) using the color spectrum. The length of the bar represents the frequency.
Hit Radius	The distance, in pixels, detectable by the Drafting Assistant between the object and the pointer.
HSL Picker	This is a standard color wheel for the Macintosh with the option to specify Hue, Saturation and Lightness.
HSV Picker	This is a standard color display for the Macintosh with the option to specify Hue, Saturation and Value.
IGES	An acronym for Initial Graphics Exchange Specification. The U.S. Department of Commerce, National Bureau of Standards issues IGES as the

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	industry standard among CAD/CAM systems for data exchange in a neutral file format.
Import	To load or read in a non-Designer Elements program file.
Infinite Plane	A two dimensional surface with no defining boundaries.
Inflection	A change in slope along a spline.
Influence	The degree of impact a control point has on a spline.
Inscribed	Within a circle. For polygons, all vertices touch the (imaginary) circle.
Instance	Refers to an object that's moved to or placed in a different location after performing an operation on it. If you create a solid, add a blend and move it, the solid is now an instance. The original is still located in the previous position although it is not displayed. If you create a master symbol, an instance occurs when you place the symbol in your drawing. An instance is associative to the original geometry. Any change made to the original is reflected in the instance. If you copy and pasted the object, the associativity is broken.
Intensity	The lighting level for a light source set in the Edit Objects dialog box.
Interference	The shared volume created by two or more intersecting objects.
Intersection	The position where two lines or curves meet. The curves may actually touch or only intersect when they are extended. The Drafting Assistant indicates only actual intersections.
ISO Lines	Control the isopram lines drawn for a surface. These Iso (isopram) lines are constant parameter curves that lie on a surface, typically defined in parameter space. The parameter space coordinate

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	system uses *U* and *V* coordinates. A 0 (zero) in both fields turns off Iso lines. The appropriate *U/V* values may enhance the visual appearance of the surface at the expense of drawing speed. The letters, *U* and *V* are industry standard space coordinate identifiers (U = horizontal, V = vertical).
Isopram	The full name for ISO Lines.
Knot Points	The points defining a spline, indicated as *vertex* points by the Drafting Assistant.
Lathe	Revolve an object a certain angle.
Layer	Analogous to transparent media used in conventional manual drafting. Parts can be constructed on several layers which you can make visible or invisible.
Line Pattern Manager	Through this dialog box you can modify the scale of all available line patterns.
Linear Duplicate	A feature that duplicates an object and places the copies in a line or in an array of multiple lines.
Links	This is the associative relationship that exists between parent/child objects in which a modification to the parent also modifies the child.
Local Face Modeling	The ability to perform various operations on a specific face of a model including: drafts, move, offset, remove replace and match.
Lock	The command to render a selection unchangeable.
Loft	A surface that references another surface making it tangent to the referenced surface. Any surface can be used to create the surface.
MxN Curves	The number of curves that define a net surface in two perpendicular directions.
Mask	To select entities as a group, masking out all others.

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Masking	The method used for applying a decal to an object, either stencil or factor.
Mate	This operation matches the planar face of one object to the planar face of another object.
Math Operator	The mathematical, trigonometric, or exponential expression used to indicate values. All value entry boxes in this Designer Elements program accept the math operators listed in Appendix A.
Max Edge	Refers to a conversion of an object type to mesh. This sets the maximum acceptable length of facets.
Message Line	The top line of the drawing area. It names the current tool and provides instructions for using it.
Memory Indicator	The indicator at the bottom left corner of the Designer Elements program window that displays how much RAM is allocated to this Designer Elements program.
Mesh	Planar elements defined by nodes or 3D vertices that can be used to represent surfaces but are not surfaces themselves.
META	The file format used by the Windows Clipboard.
Mnemonics	The key sequence which invokes a command from a menu.
Model	The model is your geometry.
Model Point	The point you can specify in the View Rotation Options dialog box about which to rotate a view.
N# Sections	The number (n) of cross sections used to define a skin surface.
Net Surface	A surface defined by M (number) of rows and N (number) of columns.
Non-planar	Surfaces or points that do not lie in a two dimensional plane.

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Normal	A perpendicular to a tangent of a curve, surface or solid face.
Normal Deviation	Refers to a conversion of an object type to mesh. This sets the maximum angular deviation between adjacent facets.
NURB	Non-Uniform Rational B-splines—the type of splines this Designer Elements program creates. NURB splines are a superset of Bezier curves. NURB splines provide designers with two interrelated functions. First, curvature continuity remains intact even when the curve is changed, so kinks won't develop as the spline is altered. Second, localized control of a complex curve is provided.
Object	An individual piece of geometry, such as a line, arc, circle, surface or solid.
Object Extents	The area defined by an object.
Object Type	This refers to a specific kind of geometry and includes curves, surfaces and solids.
Offset	The distance a curve or surface is placed from the original location.
Origin	The 0,0,0 location on the drawing area. When a new document is opened, 0,0,0 is located in the middle of the screen. The coordinate symbol displays at the origin when the grid is turned on. You can change the origin at any time.
Pan	A horizontal camera movement used when creating movies.
Palette	A group of tools. The general tool palette is always displayed to the left of the drawing area.
Parent	An object from which other objects are created or operations are performed.

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Param	The percentage point that a variable blend radius is placed along a solid edge.
Parent/Child	The relationship created between an original object and subsequent objects such that any modifications done on the parent affects the child.
Part	A collection of entities representing an object or structure.
Parting Line	A curve used with the Parting Line tool to divide a solid for applying a draft.
Paste	To place the contents of the Clipboard in the current document.
Perpendicular	At a 90° angle. The Drafting Assistant displays a notation when the current construction is at a 90° angle to an object.
Perspective	Viewing 3D geometry on a two dimensional surface as seen by normal binocular vision.
Phong Shading	Renders your geometry based on calculated light intensities at each pixel location.
Phong w/Edges Shading	Renders your geometry based on calculated light intensities at each pixel location and displays the face edge boundaries in a specific color.
Photorealistic	Images resembling photography in the rendering quality.
Pick Box	The area pixel area defined in Preferences that allows you to select an object.
Pick Objects	This is the dialog box with the Ambiguity Popup menu.
PICT	The Macintosh graphics file format which uses object-oriented bitmaps or resolution-independent graphics.
Pick	To select a location or object by clicking it.

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Pipe Solid	A solid created by with an inside and outside diameter using the Pipe Solid tool.
Pitch	The distance between helix peaks.
Planar	Surfaces or points that lie in a two dimensional plane.
Plot	To draw a part on paper using a plotter.
Plotter	A computerized drawing device for hardcopy output.
Point	A location for constructing geometry.
Point Light	This light emits rays in all directions, like a candle or table lamp.
Pointer	The position locator similar to a cursor. When a tool is in effect, the pointer takes on a representative shape while in the drawing area.
Polar Duplicate	A feature that duplicates an object and rotates the copes around a reference point.
Press	To press and hold down the mouse button. This action is most commonly used to view the contents of a menu.
Primitive	A basic solid shape defined by linear or quadratic geometry like: block, cylinder, cone, etc.
Profile	A closed curve or collection of curves connected end to end.
Protruded Feature	Material projections from a solid.
Pull-direction	The direction a part ejects from a mold.
QuickTime Object Movie	A Quick Time movie that keeps the observation point fixed as the eye point is moved at a fixed distance about the observation point.
QuickTime VR Panoramic Movie	A QuickTime movie that keeps the eye point fixed as the observation point is rotated 360°.

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Radius	Half the distance across an arc, starting from the center.
Rail	A curve used to define a sweep direction and distance for a surface.
RAW	This translator creates a file containing triangular vertices that define the x, y and z locations of the 3D faces.
Raytrace	The function that determines the visibility of objects in a scene by tracing rays from the eye to the objects. The raytrace function calculates the visibility by breaking the scene into smaller pixel areas, producing a photorealistic rendered images.
Redo	The command that restores a previous operation. You can redo an infinite number of actions in a specific Designer Elements program session.
Redraw	To refresh the screen, recreating all objects.
Reflectivity	The ratio of the amount of light falling on a material to the light reflected off it.
Relative Position	A location specified as a certain distance from another location. It is often called the delta position.
Relimit	Lengthen or shorten lines to the specified object.
Render	The command that shades your geometry in a specified mode.
Render Now	The render option activated when you choose the *Render* command. You set the mode in the Render Options dialog box.
Render Options	The dialog box that allows you to set the shading mode for your geometry.
Resolution	The appearance of surfaces and solids when rendered. You have five options: Super Fine,Very Fine, Fine, Medium and Coarse.

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Rho	Used in defining a conic object, it is the ratio of the center point - shoulder point distance and the center point - slope control point.
Right-hand rule	A memory aid for the relative directions of the positive axes. With your right palm upturned, the thumb (X) points right, the index finger (Y) points straight ahead and the middle finger (Z) points up. If you move your hand to indicate the X and Y axes, you can easily see the direction of the Z axis.
SAT	A file format for ACIS based programs.
Section View	A view created by making a section cut through your geometry displayed in a drawing view.
Selection Fence	A bounding box that temporarily appears as you drag the Selection tool cursor around an area in your drawing.
Setback	The distance that a blend extends from three or more intersecting edges.
Short Cuts	Key combinations for performing Designer Elements program operations, allowing you to customize your program.
Shelled Solid	A hollowed out solid object.
Shoulder control point	A point used in defining a conic.
Silhouette	A view of the object from the visible edges only. Silhouette edges are view dependent and can cause a significant reduction in drawing speed.
Simplify	This option available in the Stitched Solid tool asks Designer Elements program to determine whether the solid can be defined by analytic surfaces, for which ACIS is optimized, rather than nurb surfaces.
Skin Surface	Nurb patches fitted over a collection of curve cross-sections.
Slab	A solid primitive created from three or more points, a height and draft angle.

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Slope	The change of x relative to y between two points on a line. In a spline, slope defines the vector of a line tangent to the spline at a particular knot point.
Slope control point	A point used in defining a conic.
Smart Polygon	A true polygon that is one object with length and width. This can be contrasted with a single line polygon composed of individual lines.
Smart Silhouette	A silhouette that displays only if it does not degrade the performance of this Designer Elements program.
Snap	The command allows you to set the specifications for the Drafting Assistant.
Sphere Trackball	The standard trackball that allows you to rotate the view by dragging the cursor on the sphere.
Spline	A smooth, free-form curve passing through specified points.
Spot Light	This light emits a cone of light from a local source.
Static Render	This render option defines how your geometry will display when the view is stationary.
Status Line	The line at the bottom of the drawing area in which you can enter the specifications for the geometry you are creating.
Stencil	An image that defines the regions through which artwork projects onto an object.
Step Trackball	The trackball that allows you to rotate the view by clicking on a directional arrow.
Stitched Solid	This is a solid created from joining surfaces together.
STL	The file format creates stereolithography files.
Subtract Solid	This solid is subtracted from another to create a final solid.

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Sweep	Refers to the extrusion of a profile to a surface or solid.
Surface	Non Uniform Rational B-Splines created for specified boundaries.
Surface Deviation	Referring to the conversion of a surface or solid to a mesh, this sets the maximum acceptable distance between the facet and the surface represented by the mesh.
Symbol	Objects that are created in the Symbol Manager are Master Symbols and can be instanced into the drawing at multiple locations.
Tangent	The point where a line or curve touches a curve without intersecting it. The Drafting Assistant displays the *tangent* notation of a curve when the pointer nears it.
Taper Solid	This is a solid created with draft angles or tapers.
Termination Type	This defines the protrusion type used to create a feature on a solid.
3DMF	The file format of QuickDraw 3D.
Tiling	Breaks up a drawing into a multiple pages to print large drawings.
Tilt	A vertical camera movement used when creating movies.
Torus	A solid primitive generated by the revolution of a conic section, like a circle.
Transparency	The amount that light can pass through a material.
Triad	This illustrates the orientation of the x, y and z axis and the current work plane.
Toggle	To switch between two conditions, for example, *Hide Grid/Show Grid.*
Transformation	The tools to move, rotate, expand, shrink, or mirror an object or group of entities.

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Trim	To shorten or remove a portion of a line.
Tube Surface	A surface created in the shape of a tube.
Twist Angle	The angle you want your solid to twist when sweeping a profile to a solid.
Unclamped Linear	Light intensity that diminishes according to the following formula, *Intensity/distance*.
Unclamped Quadratic	Light intensity that diminishes according to the following formula, *Intensity/distance*^{2}.
Undo	The command that reverses the last editing or creation action. You can undo an infinite number of actions in a specific Designer Elements program session.
Uniform Scaling	Scales an object equally in all directions.
Units	Measures used for construction (U.S. or metric).
Vector Splines	The slope, shape, control points and control point influence can be modified for splines of this type.
Verify	A menu listing commands for determining properties of selected objects.
Vertex	The point at which the sides of an angle intersect or a knot point of a spline.
View Rotation Options	This dialog box allows you to specify trackball rotation parameters.
VRML	A file format for exporting virtual reality modeling language files.
Wrap	The mode that artwork projects onto an object, planar, cylindrical or spherical when applying a decal.
Wireframe	A 3-dimensional representation showing boundary lines, edges and intersections, but not surfaces.
World plane	The work plane used at the beginning of the construction of a model. Also known as the world coordinate system.

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Work plane	The x,y plane used for 2D objects which has an origin of 0,0,0 for all data input. Sometimes referred to as the user or work coordinate system.
Zoom	The tool or command that allows you to magnify or reduce an image.

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